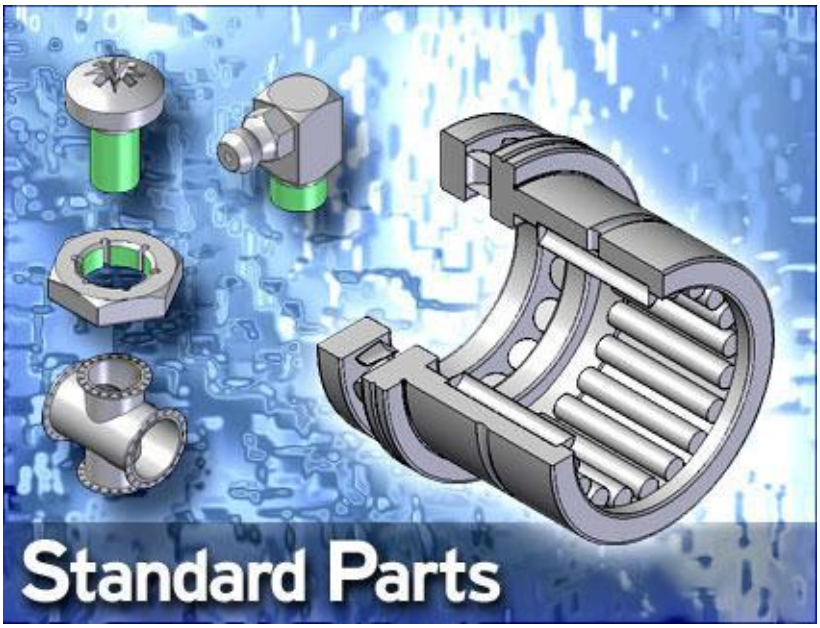
Standard Parts ST2

User Guide



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Document Scope

This document provides a detailed description of Solid Edge Standard Parts extensions (add-on) for the Solid Edge application. This document should be used by Solid Edge Standard Parts users and administrators.

Solid Edge Standard Parts

Solid Edge Standard Parts is an add-on application for the Siemens 3D CAD application, Solid Edge. Solid Edge Standard Parts is a powerful parts management system that allows designers to define, store, select and position commonly used parts – fasteners, bearings, structural members, pipes and fittings – quickly and efficiently, enabling rapid and precise completion of 3D assemblies. With Solid Edge Standard Parts, companies can establish and share their own standards, so designers can concentrate on creative design and not on redundant modeling tasks.

Basic Definitions

For more details about Solid Edge/Insight/Teamcenter installation and workflows, refer to the Solid Edge/Insight/Teamcenter documentation. Some Standard Parts themes are described in the Solid Edge help file.

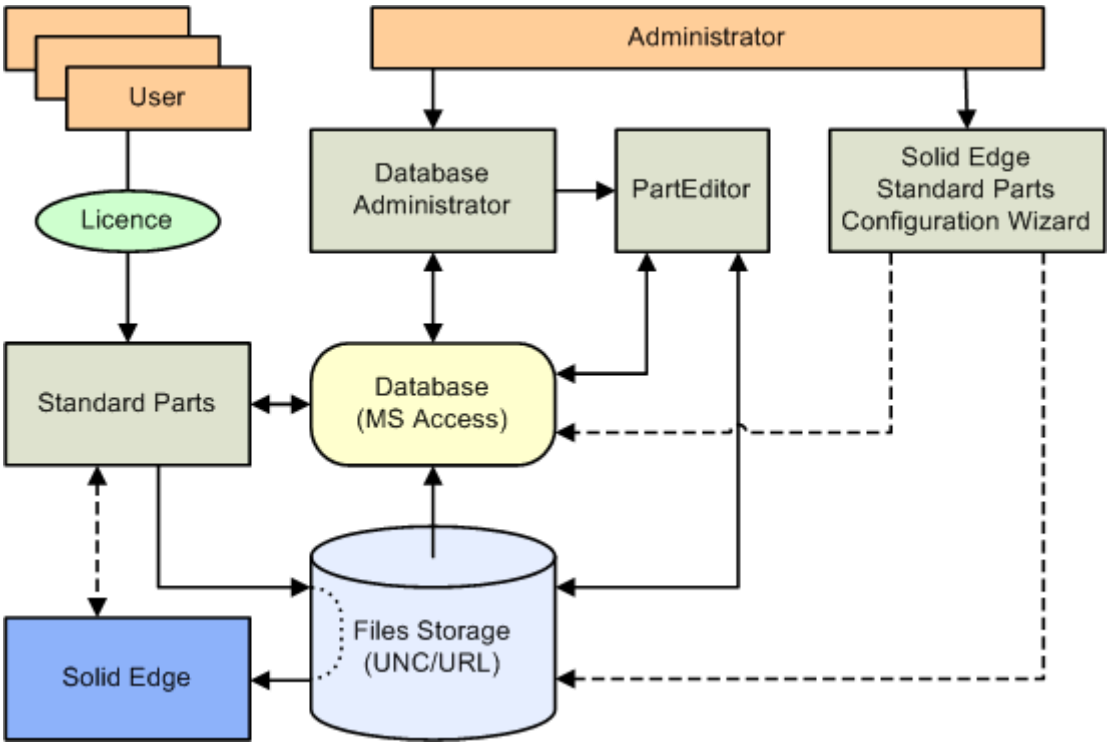
- **Standard Parts** – main application. Allows a user to find and place desirable parts from the Standard Parts database. For more details, see its description below.
- **Solid Edge Standard Parts Administrator** – application for database administration. For more details see its description below.
- **Database** – all Standard Parts documents information storage. For more details, see the Database and Solid Edge Standard Parts Administrator description below.
- **Solid Edge Standard Parts Configuration Wizard** – application for managing Solid Edge Standard Parts global configuration. For more details, see its description below
- **BulkPartAdder** – application for creating Customer Standard Parts from existing Solid Edge documents. BulkPartAdder is a part of Solid Edge Standard Parts Administrator.
- **SmartInstaller** – application for installing and updating Standard Parts within working database. SmartInstaller is a part of Solid Edge Standard Parts Administrator.
- **PartEditor** – application for changing configuration and adding the Standard and Customer Standard Parts
- **Standard Parts Installation Database** - *.ddb file containing information on Standard Parts in delivered Standard Parts Library. This information serves mainly to keep the parts registered in the database up-to-date. That means, that if a new version of Standard Parts Libraries will be available, only parts which have changes (compared to older versions) may be updated in working database.
- **Solid Edge Machinery and Piping Library** – Library of international norms of standard parts prepared by Siemens; contains ANSI, DIN, UNI, GB, JIS, GOST, CSN norms in Solid Edge ST.
- **Standard Parts installation folder** – main (usually root) folder of the Standard Parts system. This folder contains subfolders with Library parts, configuration files and others. For more details, see a description of Solid Edge Standard Parts Configuration Wizard.
- **Standard Parts folder** – folder with your standard parts. For more details, see the description of Solid Edge Standard Parts Configuration Wizard and Standard Parts.
- **Standard Part** – Siemens Library standard part. It can be a Parent part or Single part.

- **Customer Standard Part** – Customer specific Standard Part. It can be prepared using BulkPartAdder or Part Editor. For more details see its specification below.
- **Parent part** – Siemens Library Standard global file. Contains definitions of all relevant Single parts. It is delivered with Standard Parts Library to a version specific folder.
- **Single part** – generated final part. Derived from Parent part.
- **Part name** – global part identifier. It should be a full, detailed and unique descriptor for every file in the Standard Parts database. It serves for fast and correct identification of parts in Standard Parts applications.
- **SAC file** (SMAPINIFile.sac) – global definition and settings file.

Solid Edge Standard Parts Scheme

Solid Edge Standard Parts is a simple database oriented document management system for working with standardized parts. The most important user application is Standard Parts. This application allows users to find, select and place (eventually replace or modify) centralized and standardized parts into assemblies. The most common use is to select standard parts from the Solid Edge Standard Parts Library or Customer Standard Parts and place into an assembled project.

Administrator applications are called: Solid Edge Standard Parts Configuration Wizard, Solid Edge Standard Parts Administrator and PartEditor. The Solid Edge Standard Parts Configuration Wizard allows system administrators to define a document management structure, paths and databases for all users. The Solid Edge Standard Parts Administrator (including integrated SmartInstaller and so called BulkPartAdder) is an application that manages database and database content. The administrator can delete or add desirable standardized parts into a database. The last administrative application is PartEditor; this application allows managing and changing the content (characteristics, properties etc.) of every standard part.

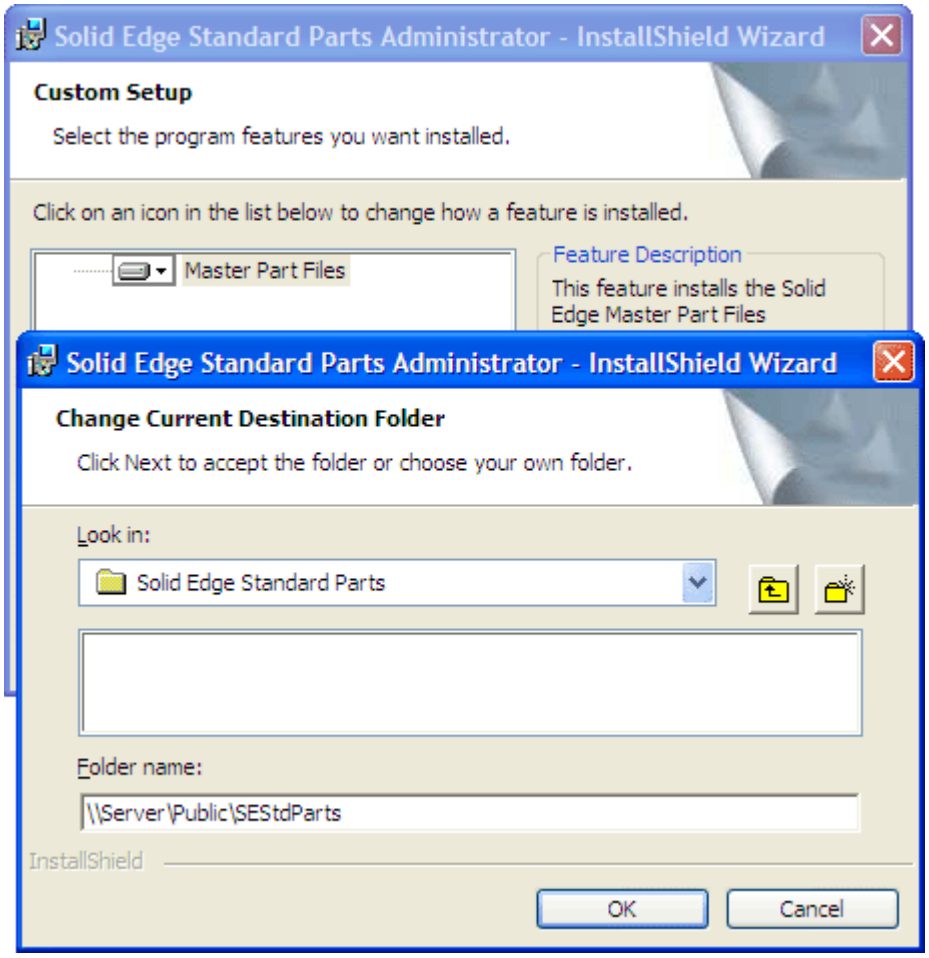


Solid Edge Standard Parts Installation (Local or Network)

Standard Parts Administration Installation

1. Install Solid Edge on all workstations.

2. Install "Standard Parts Server" on the workstation used for Standard Parts administration. Change the destination folder to a local/network location accessible from other workstations[1] (e.g. "\\Server\Public\SEStdParts" as shown in figure bellow). Full access permissions are required for the target location.


Solid Edge Standard Parts Administrator - InstallShield Wizard
Custom Setup
Select the program features you want installed.
Click on an icon in the list below to change how a feature is installed.
Master Part Files
Feature Description
This feature installs the Solid Edge Master Part Files
Solid Edge Standard Parts Administrator - InstallShield Wizard
Change Current Destination Folder
Click Next to accept the folder or choose your own folder.
Look in:
Solid Edge Standard Parts
Folder name:
\\Server\Public\SEStdParts
InstallShield
OK
Cancel

3. Complete the installation.

[1] It is currently possible and preferred to identify the destination folder by typing the UNC address, rather than by browsing in "Look in"! Nevertheless, local paths or mapped disks are supported.

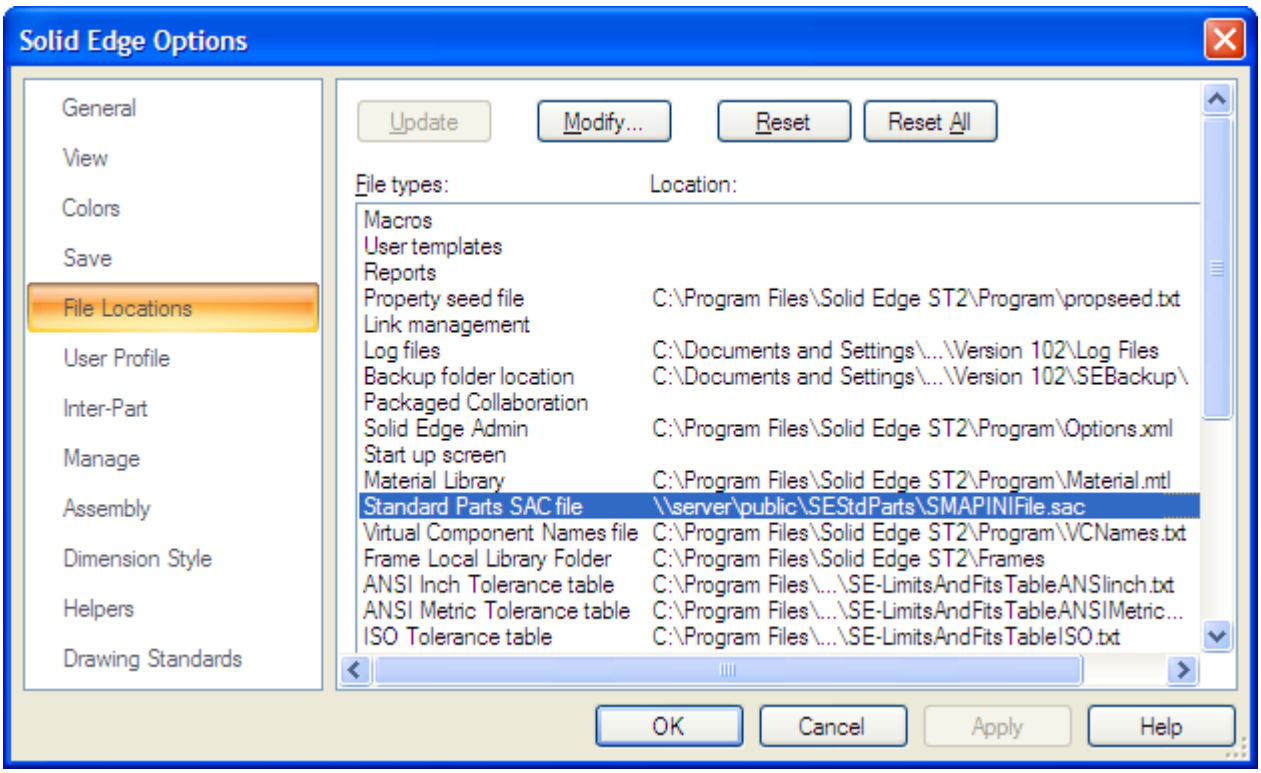
After completing these steps, the Solid Edge Standard Parts installation folder structure is created (in this case at the location "\\Server\Public\SEStdParts"). For more details about folder structure, see the section "Solid Edge Standard Parts Configuration Wizard". It contains:

- "Database", a folder containing Standard Parts database "StandardParts102" (as default) with Free Content Standard Parts version 8.95 registered.
- "Html" folder
- "Standard Parts" folder containing Free Content parent parts (Standard Parts\3D-Standard_Parts_V8.95\Parent_Parts)
- "Working folder"
- "SMAPINIFile.sac"

Note: If the Master Part Files (Free Content) is not installed in Step 2, the folder structure described above is not created. To create it, some Standard Parts Library (Machinery or Piping) must be installed or Solid Edge Standard Parts Configuration Wizard must be used.

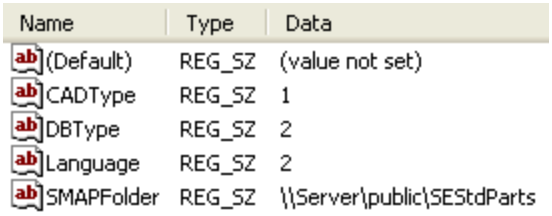
Standard Parts Workstation Installation

If the administration workstation is set up; on other workstations just set the Standard Parts SAC file location (Solid Edge Options – File Locations). The SAC file must be located in the UNC path which has been set during the Standard Parts administration installation (step 2). For the URL location, see the "Insight Installation" section.



Registry Content

For the correct functionality of the Solid Edge Standard Parts system, the Standard Parts registry key ("HKEY_LOCAL_MACHINE\SOFTWARE\Unigraphics Solutions\Standard Parts" for 32 bit or "HKEY_LOCAL_MACHINE\SOFTWARE\Wow6432Node\Unigraphics Solutions\Standard Parts" for 64 bit operating systems) must contain the following string values:



- CADType – this value reflects the actual state of Solid Edge. If it is installed correctly, "1" must be set.
- DBType – type of the database. It must always be set to "2".
- Language – current user interface language. Available values are listed in the table below.
- SMAPFolder – location of the top level Standard Parts folder. This value is set during installation or after saving settings in Solid Edge Standard Parts Configuration Wizard.

Value	Language
1	German
2	English
3	French
4	Italian
5	Spanish
6	Russian
7	Japanese
8	Korean
9	Chinese Traditional
10	Chinese Simplified
11	Hungarian
12	Polish
13	Czech

Note: Standard Parts database is language dependent. After the user interface language is changed, you should set the database created for the selected language as a working database or create a new one (the only exemption is a change from any language to English). This is possible only in Solid Edge Standard Parts Configuration Wizard.

Installing Libraries and Registering Standard Parts

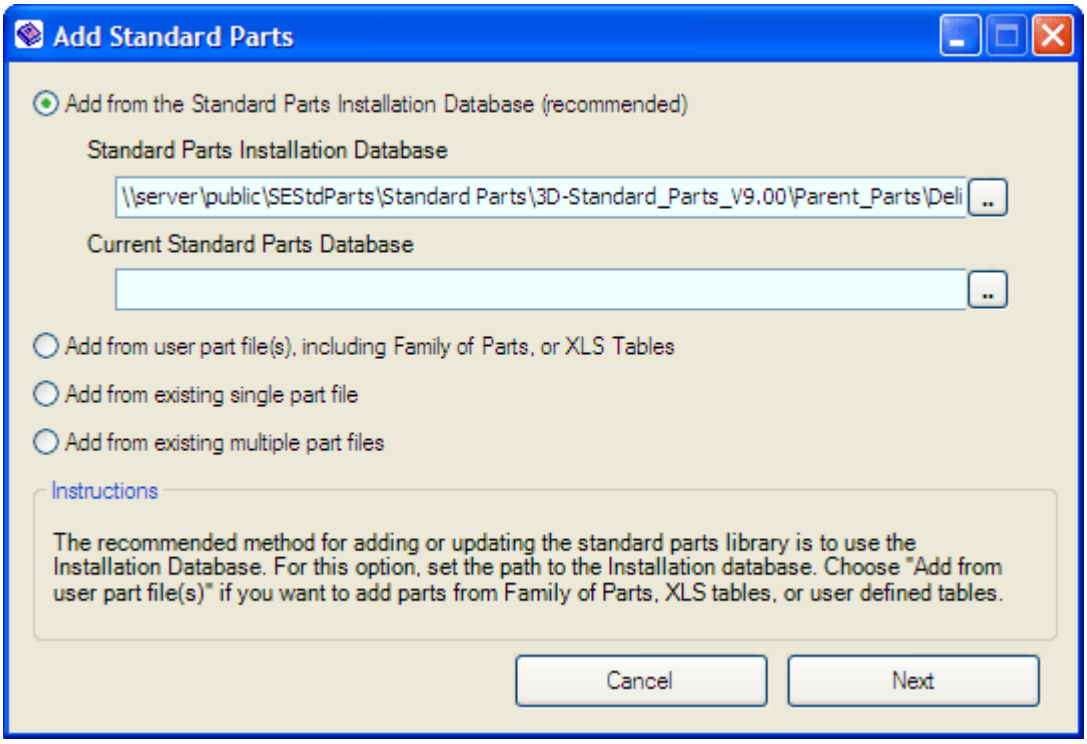
1. If you have purchased Standard Parts libraries, install them directly at the top level of the Solid Edge Standard Parts installation directory. The installation process will automatically copy all standard parts from the CD/DVD to a specific location of the Standard Parts folder. When you are installing the library, it is important to select the top level folder created in step 2 of "Standard Parts administration"!!!

2. When the Library installation is complete, launch Solid Edge Standard Parts Administrator and click on "**Add parts**".

Registering Standard Parts in Database via Add Parts Button in Solid Edge Standard Parts Administrator

Add Parts Options

1. **Add from the Standard Parts Installation Database** – launches the SmartInstaller application. Default option (see the Registering Standard Parts Using Installation Database chapter for details).
2. **Add from user part file(s), including Family of Parts, or XLS Tables** – launches the BulkPartAdder application for registering user documents as Standard Parts (see the Creating Customer Standard Parts chapter for details).
3. **Add from existing single part file** – registers selected single part (or multipart) in the database (see the Adding Existing Single Part File chapter for details).
4. **Add from existing multiple part files** – registers all standard parts from a selected location (see the Adding Existing Part Files chapter for details).


Add Standard Parts
Add from the Standard Parts Installation Database (recommended)
Standard Parts Installation Database
\\server\public\SEStdParts\Standard Parts\3D-Standard_Parts_V9.00\Parent_Parts\Deli
..
Current Standard Parts Database
..
Add from user part file(s), including Family of Parts, or XLS Tables
Add from existing single part file
Add from existing multiple part files
Instructions
The recommended method for adding or updating the standard parts library is to use the Installation Database. For this option, set the path to the Installation database. Choose "Add from user part file(s)" if you want to add parts from Family of Parts, XLS tables, or user defined tables.
Cancel
Next

Insight Installation

This section describes how to migrate (fully or partially) your local or network system (located at UNC path) to SharePoint (URL location). Direct installation/configuration with SharePoint from Standard Parts setup is not possible with this version.
All Insight (server/client) installation requirements must be met prior to performing these migration steps. For more details about installation and working with Insight, see Microsoft and Siemens documentation for SharePoint and Insight services (Insight Implementation Guide). The minimum supported role at SharePoint is "Contributor" for all users.
The Solid Edge Standard Parts system has many limitations for Insight localization that are derived from SharePoint and Insight limitations and rules; most important are:

1. Global limitations:
    a. There is a limitation for the filename/path length. Also, some special characters are not supported.
    b. Mandatory Insight properties must be added to all Document Libraries using the tool "Workspace Editor" at the server (pass full URLs, i.e. http://server/doclib) prior to importing files to the library.

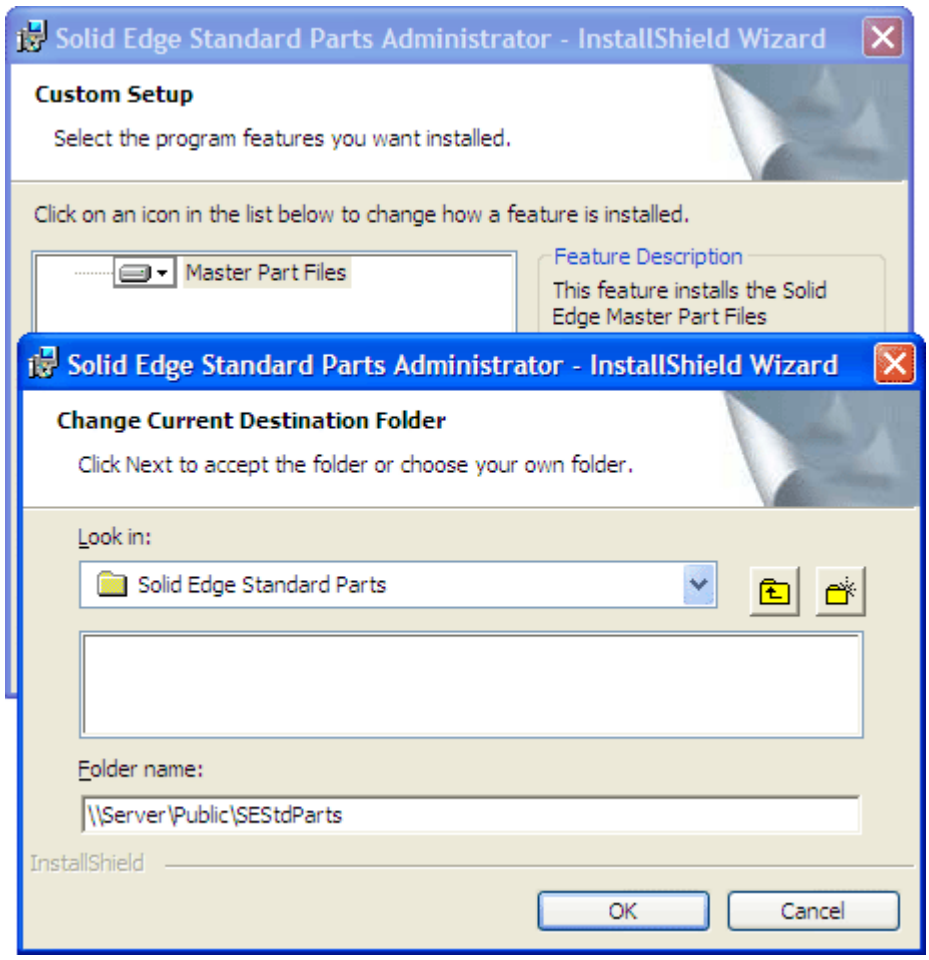
2. Workstation limitations:
    a. Web Client service must be disabled (to correctly add a Network Place on your workstation pointing to the SharePoint URL).
    b. Proxy must be disabled or bypassed for LAN!

Insight modes

Solid Edge ST allows use of Insight in two different modes.
The first one – **"All on Insight" mode -** requires that the whole Standard Parts folder structure (including the SAC file and excluding the database) resides on Insight.
The second one – **"Generated parts and working folder on Insight" mode** – uses Insight only as a location for generated parts and working folder.

Standard Parts Administration Installation ("All on Insight" mode)

1. Install Solid Edge on all workstations.
2. Install "Standard Parts Server" on the workstation you want to use for remote Standard Parts administration[1]. When asked for the location, change the destination folder to some UNC location that is accessible from this workstation (e.g. "\\Server\Public\SEStdParts"; see the picture bellow).
3. Complete the installation.
4. If you need to install the Standard Parts full version of Libraries, you can do it in the same method described in the section above: "Installing Libraries" (but you don't need to register your parts in the database).
5. Create a target SharePoint document library (http://is18/sites/example/StdPartsST2 in our case). Do not forget to add Insight mandatory properties from Insight Server at this document library.

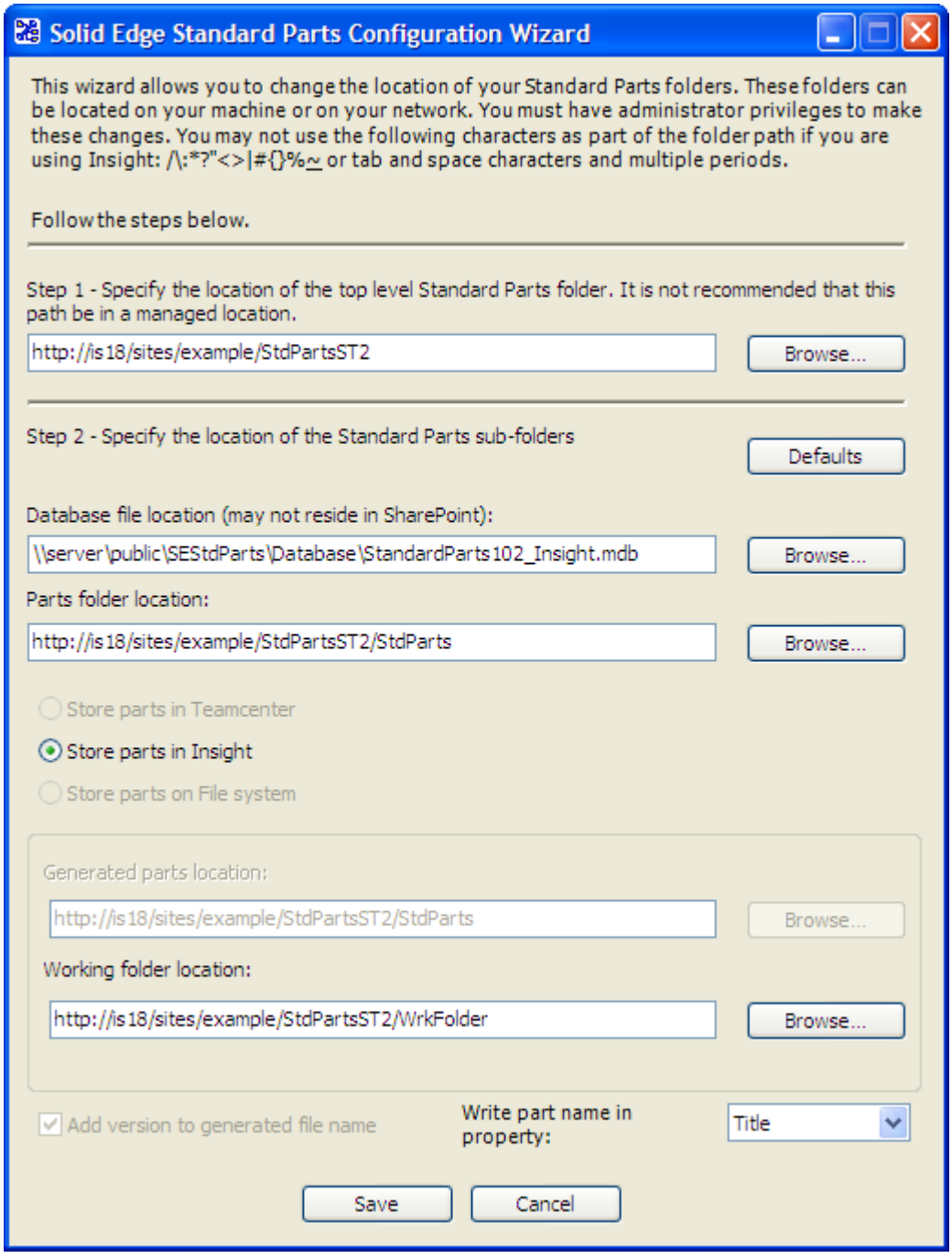

Solid Edge Standard Parts Administrator - InstallShield Wizard
Custom Setup
Select the program features you want installed.
Click on an icon in the list below to change how a feature is installed.
Master Part Files
Feature Description
This feature installs the Solid Edge Master Part Files
Solid Edge Standard Parts Administrator - InstallShield Wizard
Change Current Destination Folder
Click Next to accept the folder or choose your own folder.
Look in:
Solid Edge Standard Parts
Folder name:
\\Server\Public\SEStdParts
InstallShield
OK
Cancel

Standard Parts administration installation (Step 2).

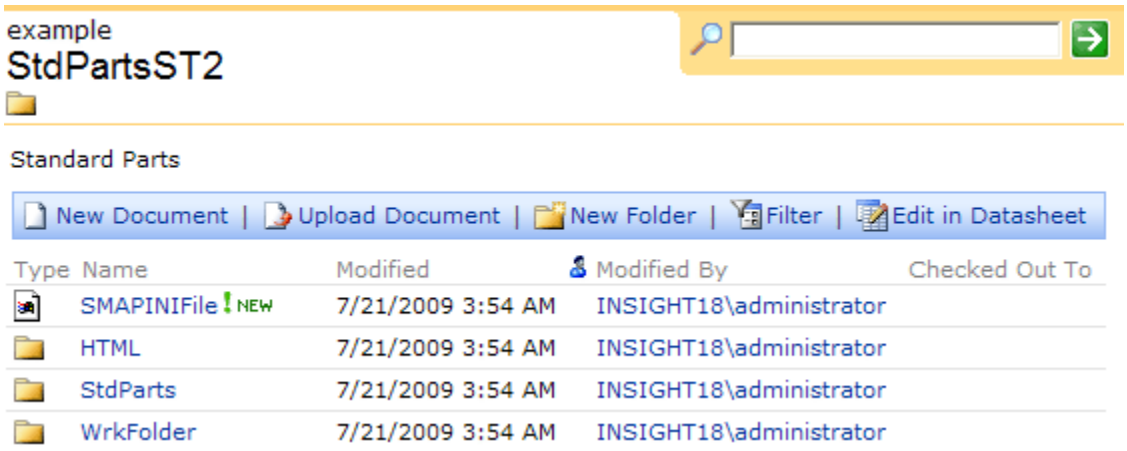
6. Add a Network Place on your workstation pointing to the SharePoint URL. This step is necessary for accessibility from some standard dialogs not customized for working with URLs.

[1] Full Solid Edge software must be installed. Solid Edge Insight Server or Insight Connect installation is not supported!!!

7. Launch the Configuration Wizard
    a. In "Step 1" of the Configuration Wizard "Browse" to the Network Place created in previous step.
    b. Click on "Defaults" (default folder structure will be displayed; see the picture below).
    c. Create a new database at some desirable network location accessible to all users (typically the same as in Step 2).
    d. Click on "Save". The Solid Edge Standard Parts directory structure will be created at the SharePoint location.


Solid Edge Standard Parts Configuration Wizard
This wizard allows you to change the location of your Standard Parts folders. These folders can be located on your machine or on your network. You must have administrator privileges to make these changes. You may not use the following characters as part of the folder path if you are using Insight: /\:*?"<>|#{}%~ or tab and space characters and multiple periods.
Follow the steps below.
Step 1 - Specify the location of the top level Standard Parts folder. It is not recommended that this path be in a managed location.
http://is18/sites/example/StdPartsST2
Browse...
Step 2 - Specify the location of the Standard Parts sub-folders
Defaults
Database file location (may not reside in SharePoint):
\\server\public\SEStdParts\Database\StandardParts102_Insight.mdb
Browse...
Parts folder location:
http://is18/sites/example/StdPartsST2/StdParts
Browse...
Store parts in Teamcenter
Store parts in Insight
Store parts on File system
Generated parts location:
http://is18/sites/example/StdPartsST2/StdParts
Browse...
Working folder location:
http://is18/sites/example/StdPartsST2/WrkFolder
Browse...
Add version to generated file name
Write part name in property:
Title
Save
Cancel

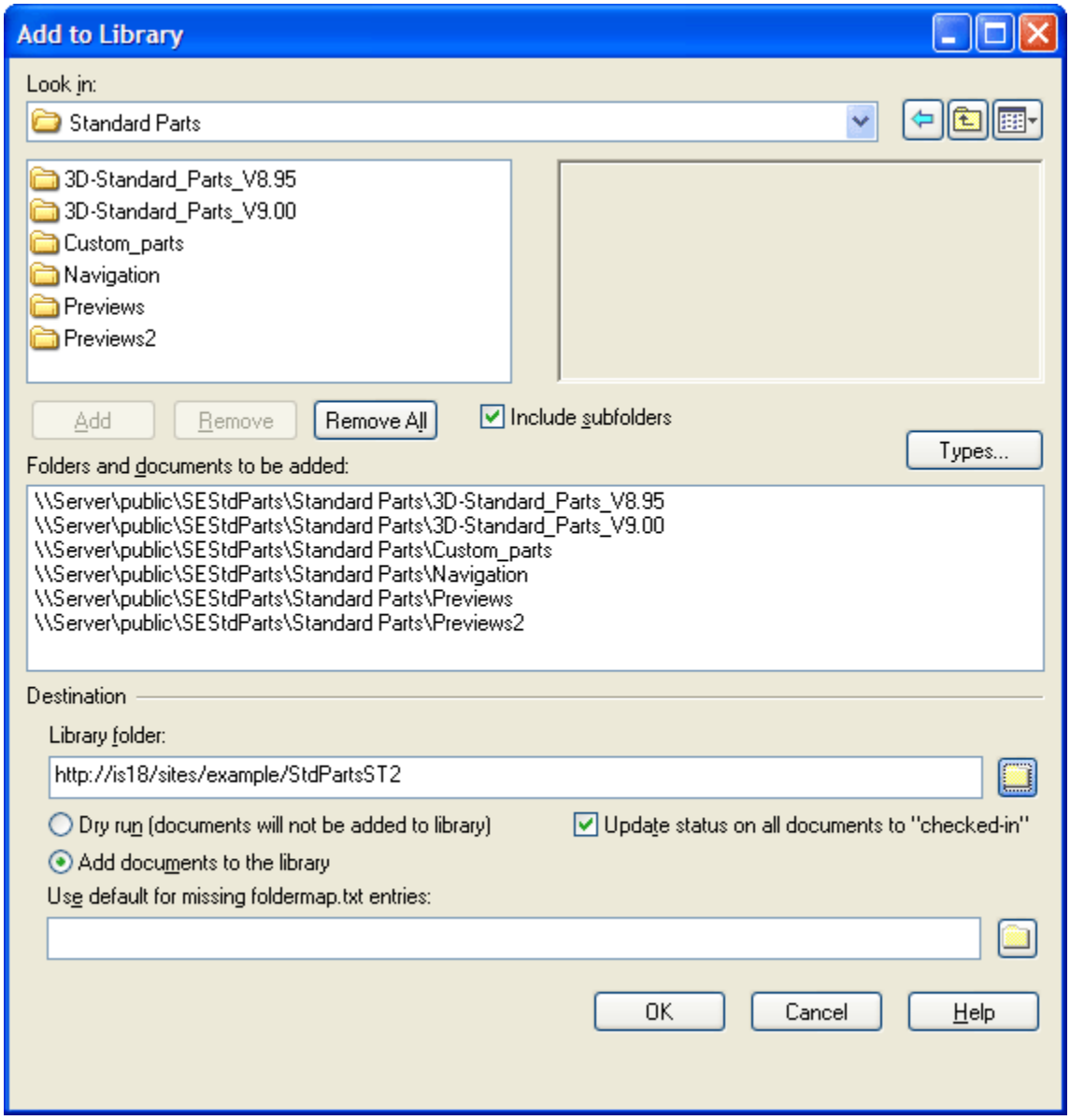
Standard Parts settings in Configuration Wizard (Step 7).


example
StdPartsST2
Standard Parts
New Document | Upload Document | New Folder | Filter | Edit in Datasheet
Type Name Modified Modified By Checked Out To
SMAPINIFile NEW 7/21/2009 3:54 AM INSIGHT18\administrator
HTML 7/21/2009 3:54 AM INSIGHT18\administrator
StdParts 7/21/2009 3:54 AM INSIGHT18\administrator
WrkFolder 7/21/2009 3:54 AM INSIGHT18\administrator

SharePoint document library created in Step 5 with the SAC file and folder structure created in Step 7.

8. Add the Document Library location ("http://is18/sites/example/StdPartsST2" in our case) to the "searchscope.txt" file (typically located at "C:\Program Files\Solid Edge ST2\Program").

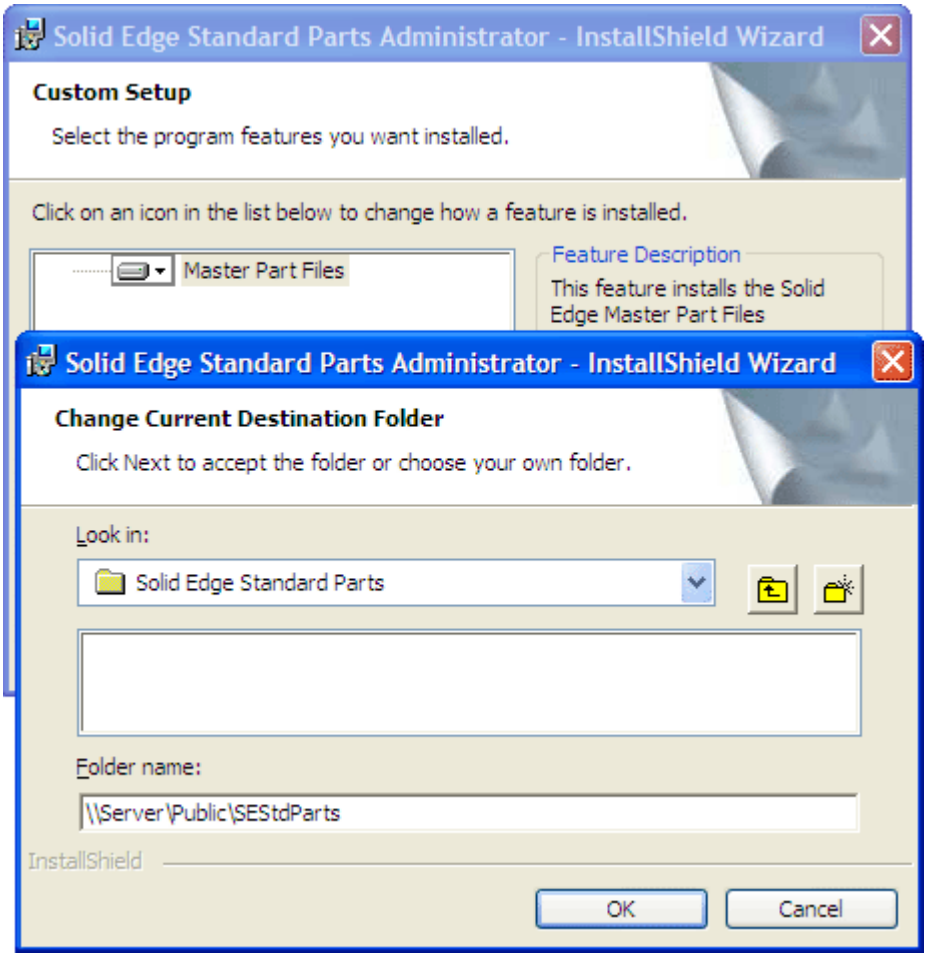
9. Launch Insight Connect (View and Markup) and add all subfolders of the Standard Parts folder (3D-Standard_Parts_Vx.xx, Previews and Navigation) from a pre-prepared UNC installation ("\\Server\Public\SEStdParts\Standard Parts") to the location of the Standard Parts library ("http://is18/sites/example/StdPartsST2"). This action can take several minutes (depending on the file size and number of files that you are attempting to upload).



10. Launch DB Administrator to add your parts into database.

Standard Parts Administration Installation ("Generated parts and working folder on Insight" mode)

1. Install Solid Edge on all workstations.
2. Install Standard Parts on the workstation you want to use for remote Standard Parts administration[1]. When asked for the location, change the destination folder to some UNC location that is accessible from this workstation (e.g. "\\Server\Public\SEStdParts"; see the picture bellow).
3. Complete the installation.
4. If you need to install the Standard Parts full version of Libraries, you can do it using the same method described in the section above: "Installing Libraries" (but you don't need to register your parts with the database).
5. Create a target SharePoint document library (http://is18/Example/StdPartsST2 in our case). Do not forget to add Insight mandatory properties from Insight Server at this document library.

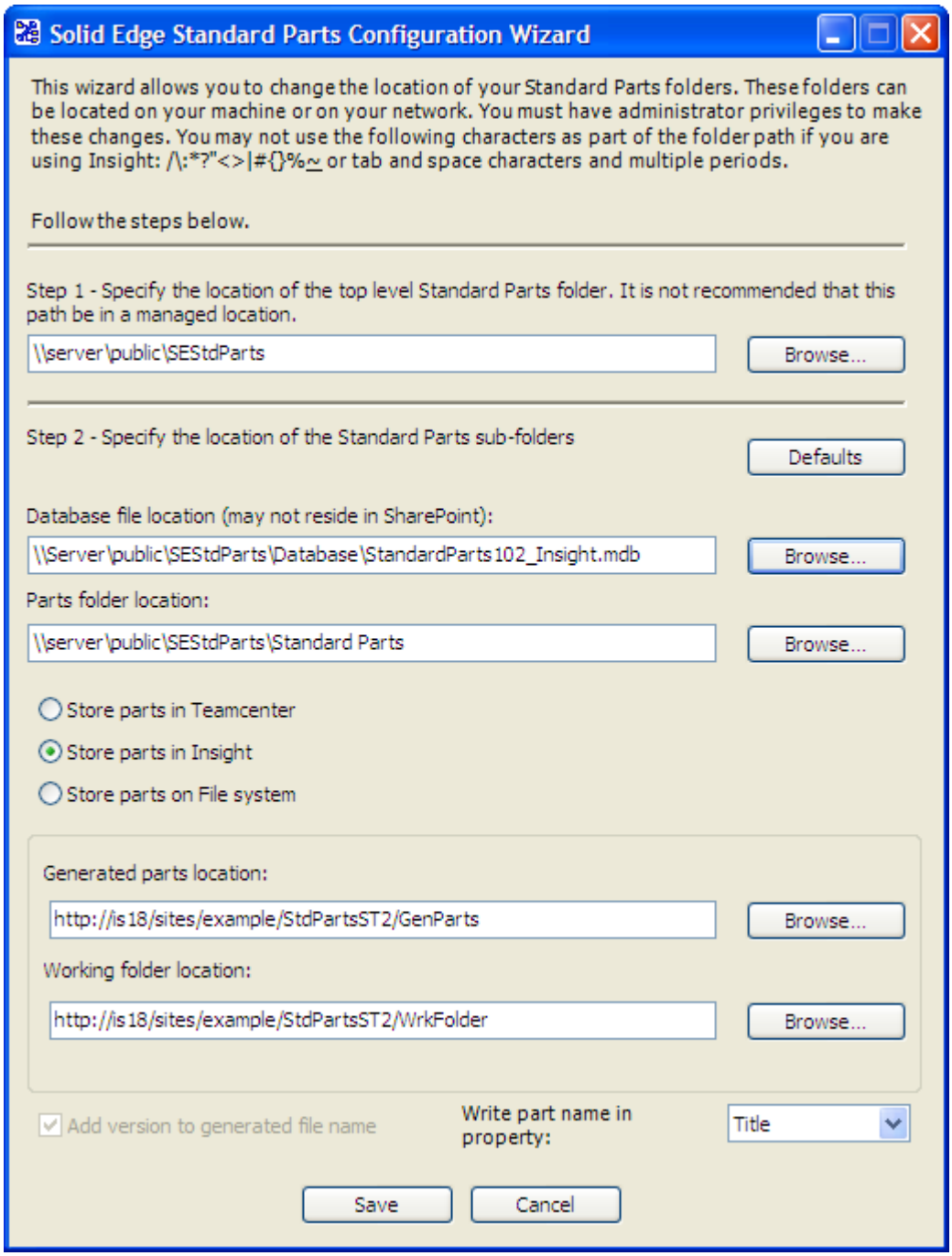

Solid Edge Standard Parts Administrator - InstallShield Wizard
Custom Setup
Select the program features you want installed.
Click on an icon in the list below to change how a feature is installed.
Master Part Files
Feature Description
This feature installs the Solid Edge Master Part Files
Solid Edge Standard Parts Administrator - InstallShield Wizard
Change Current Destination Folder
Click Next to accept the folder or choose your own folder.
Look in:
Solid Edge Standard Parts
Folder name:
\\Server\Public\SEStdParts
InstallShield
OK
Cancel

Standard Parts administration installation (Step 2).

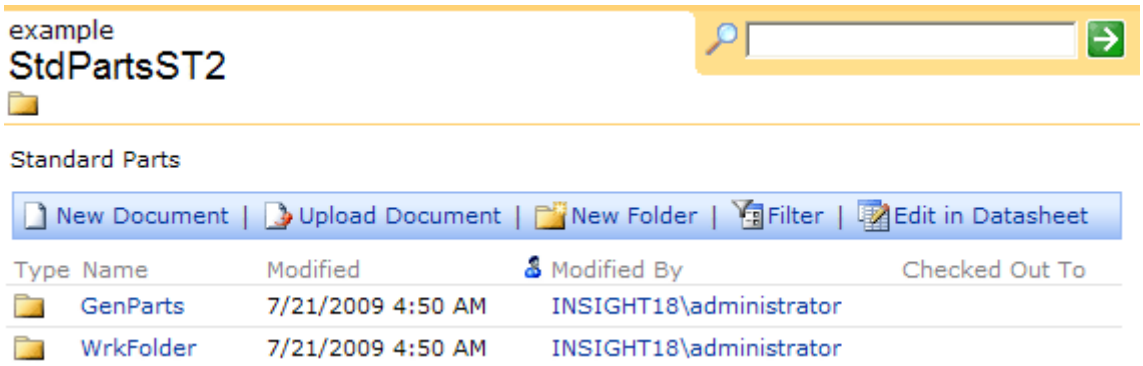
6. Add a Network Place on your workstation pointing to the SharePoint URL. This step is necessary for accessibility from some standard dialogs not customized for working with URLs.

[1] Full Solid Edge software must be installed. Solid Edge Insight Server or Insight Connect installation is not supported!!!

7. Launch the Configuration Wizard
    a. Select **Store parts in Insight**.
    b. Select a SharePoint location for **generated parts** (http://is18/Example/StdPartsST/GenParts in our case).
    c. Select a SharePoint location for the **working folder** (http://is18/Example/StdPartsST/WrkFolder in our case).
    d. Click on "Save" to confirm the settings.


Solid Edge Standard Parts Configuration Wizard
This wizard allows you to change the location of your Standard Parts folders. These folders can be located on your machine or on your network. You must have administrator privileges to make these changes. You may not use the following characters as part of the folder path if you are using Insight: /\:*?"<>|#{}%~ or tab and space characters and multiple periods.
Follow the steps below.
Step 1 - Specify the location of the top level Standard Parts folder. It is not recommended that this path be in a managed location.
\\server\public\SEStdParts
Browse...
Step 2 - Specify the location of the Standard Parts sub-folders
Defaults
Database file location (may not reside in SharePoint):
\\Server\public\SEStdParts\Database\StandardParts102_Insight.mdb
Browse...
Parts folder location:
\\server\public\SEStdParts\Standard Parts
Browse...
Store parts in Teamcenter
Store parts in Insight
Store parts on File system
Generated parts location:
http://is18/sites/example/StdPartsST2/GenParts
Browse...
Working folder location:
http://is18/sites/example/StdPartsST2/WrkFolder
Browse...
Add version to generated file name
Write part name in property:
Title
Save
Cancel

Standard Parts settings in Configuration Wizard (Step 7).

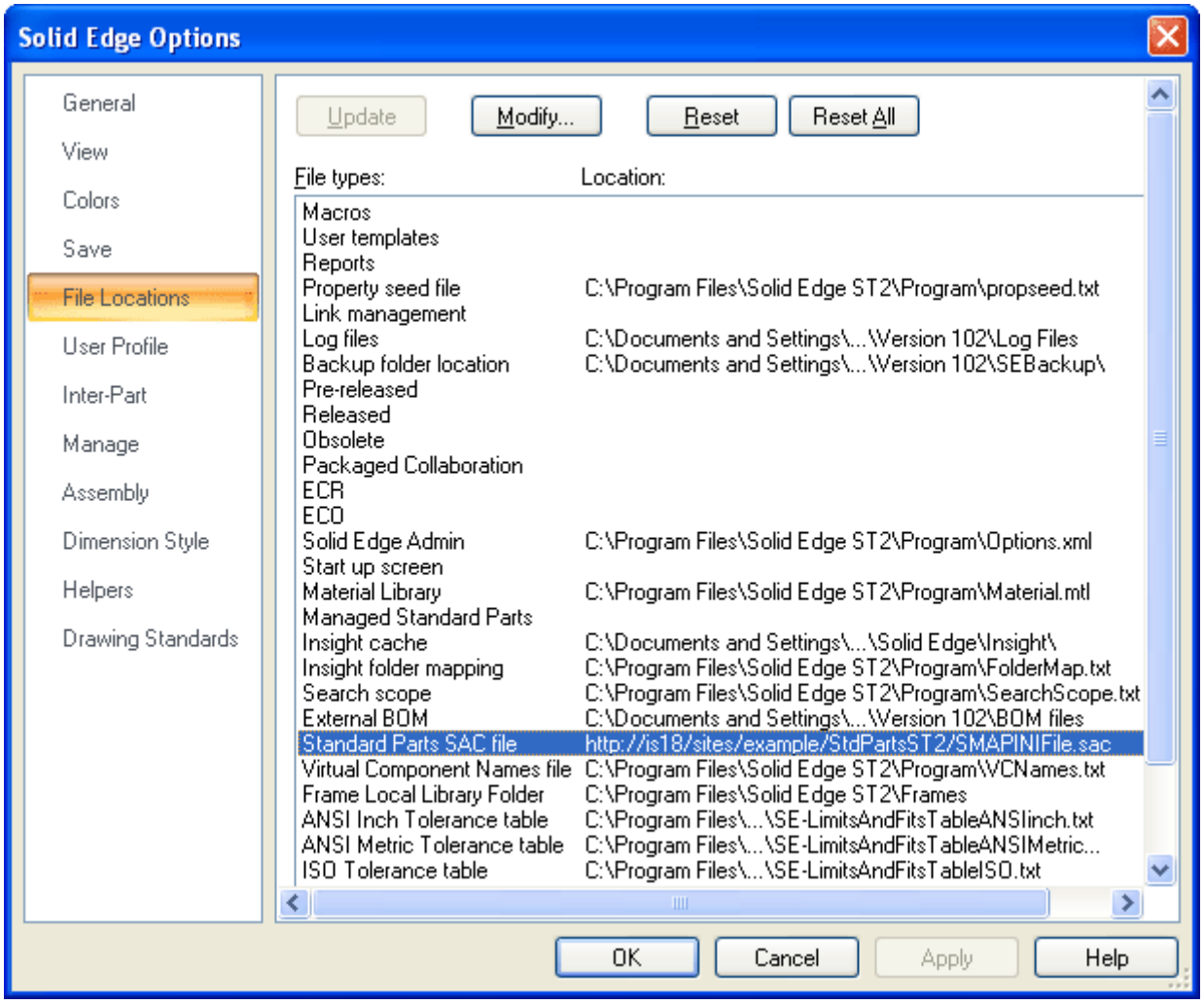

example
StdPartsST2
Standard Parts
New Document | Upload Document | New Folder | Filter | Edit in Datasheet
Type Name Modified Modified By Checked Out To
GenParts 7/21/2009 4:50 AM INSIGHT18\administrator
WrkFolder 7/21/2009 4:50 AM INSIGHT18\administrator

SharePoint document library created in Step 5 with the folders structure created in Step 7.

8. Add the Document Library location ("http://is18/sites/example/StdPartsST2" in our case) to the "searchscope.txt" file (typically located at "C:\Program Files\Solid Edge ST2\Program").
9. Launch DB Administrator to add your parts into database.

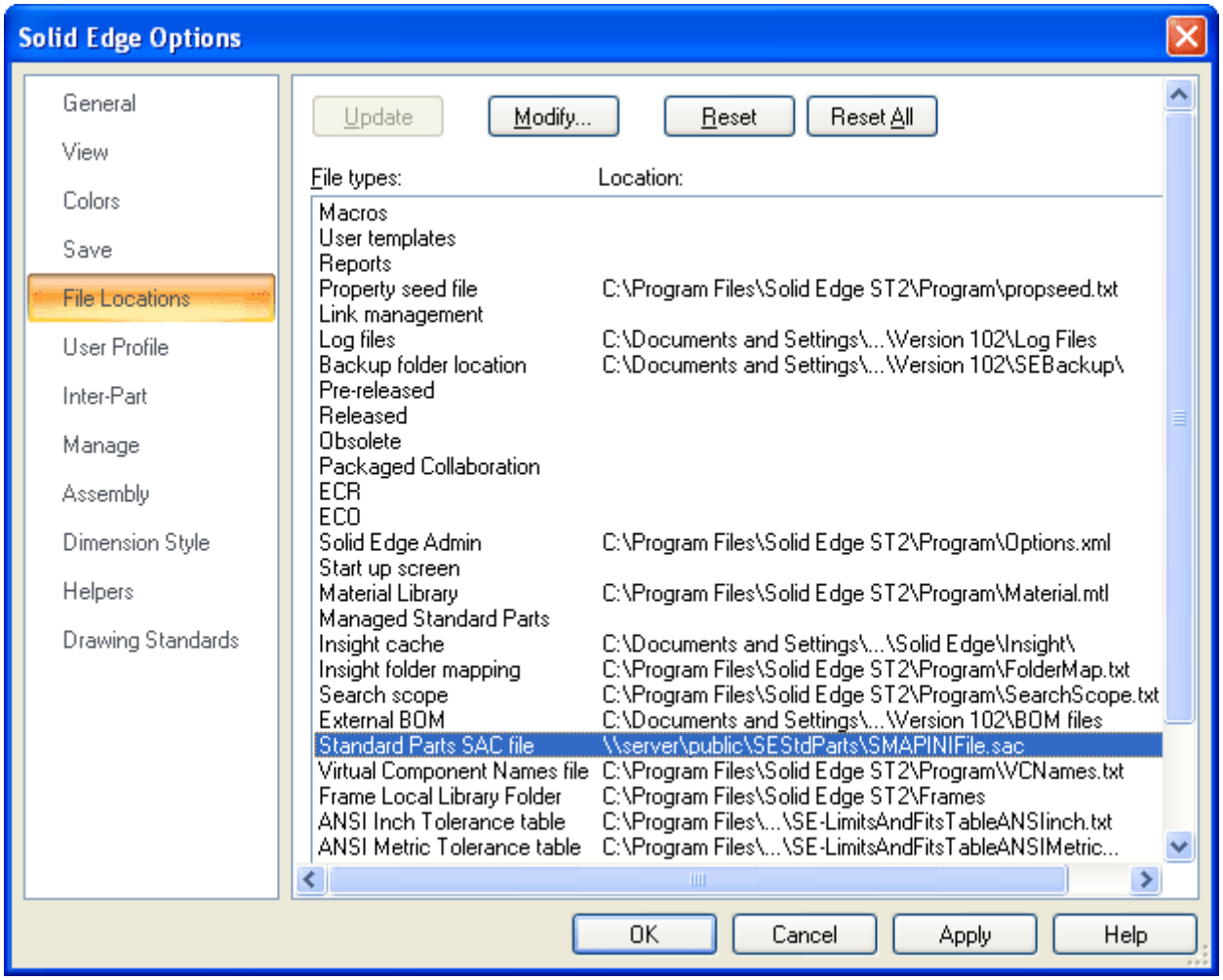
Standard Parts Workstation Installation ("All on Insight" mode)

If the administrative workstation has been set up; on other workstations, set the Standard Parts SAC file location (Tools – Options – File Locations) to the appropriate SharePoint URL (http://is18/sites/example/StdPartsST2/SMAPINIFile.sac in this case). If you are using a different search-scope.txt than the administrative workstation, do not forget to add the target document library ("http://is18/sites/example/StdPartsST2") to this file. Do not forget to configure SharePoint / Insight local settings as described for the administrative workstation (disable web client service, etc.).



Standard Parts Workstation Installation ("Generated parts and working folder on Insight" mode)

If the administration workstation is set up; on other workstations just set the Standard Parts SAC file location (Tools – Options – File Locations), the same as for a local or network installation. The SAC file must be located in the same UNC folder that was set during the Standard Parts administration installation. The document library containing the folder for generated parts and the working folder must be set in the "searchscope.txt" file.

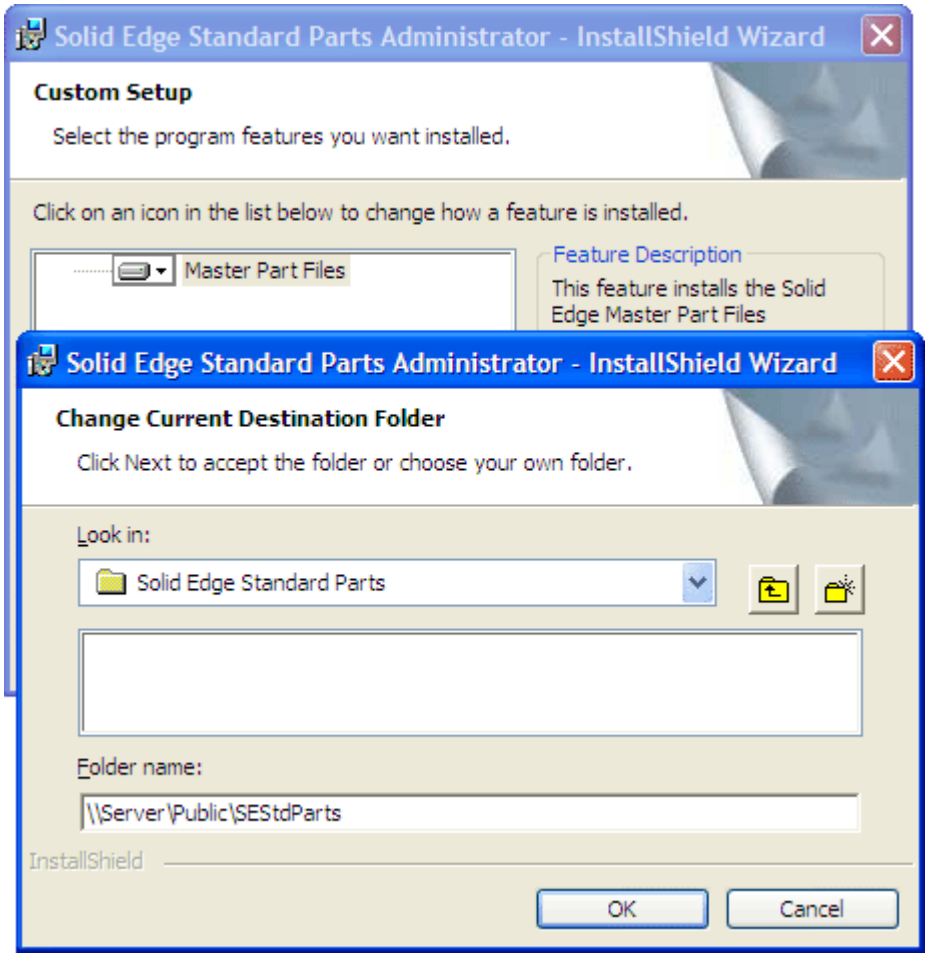


Teamcenter Installation

This section describes how to set your local or network system (located on a UNC path) to store generated parts and newly created documents in Teamcenter.
All Teamcenter (server/client) installation requirements must be met prior to performing these installation steps. For more details about installation and working with Teamcenter, see corresponding documentation.

Standard Parts Administration Installation (Teamcenter)

1. Install Teamcenter client, Solid Edge and Solid Edge Embedded Client on all workstations.
2. Install Standard Parts on the workstation you want to use for remote Standard Parts administration[1]. When asked for the location, change the destination folder to a UNC folder that is accessible from this workstation (e.g. "\\Server\Public\SEStdParts"; see the picture below).

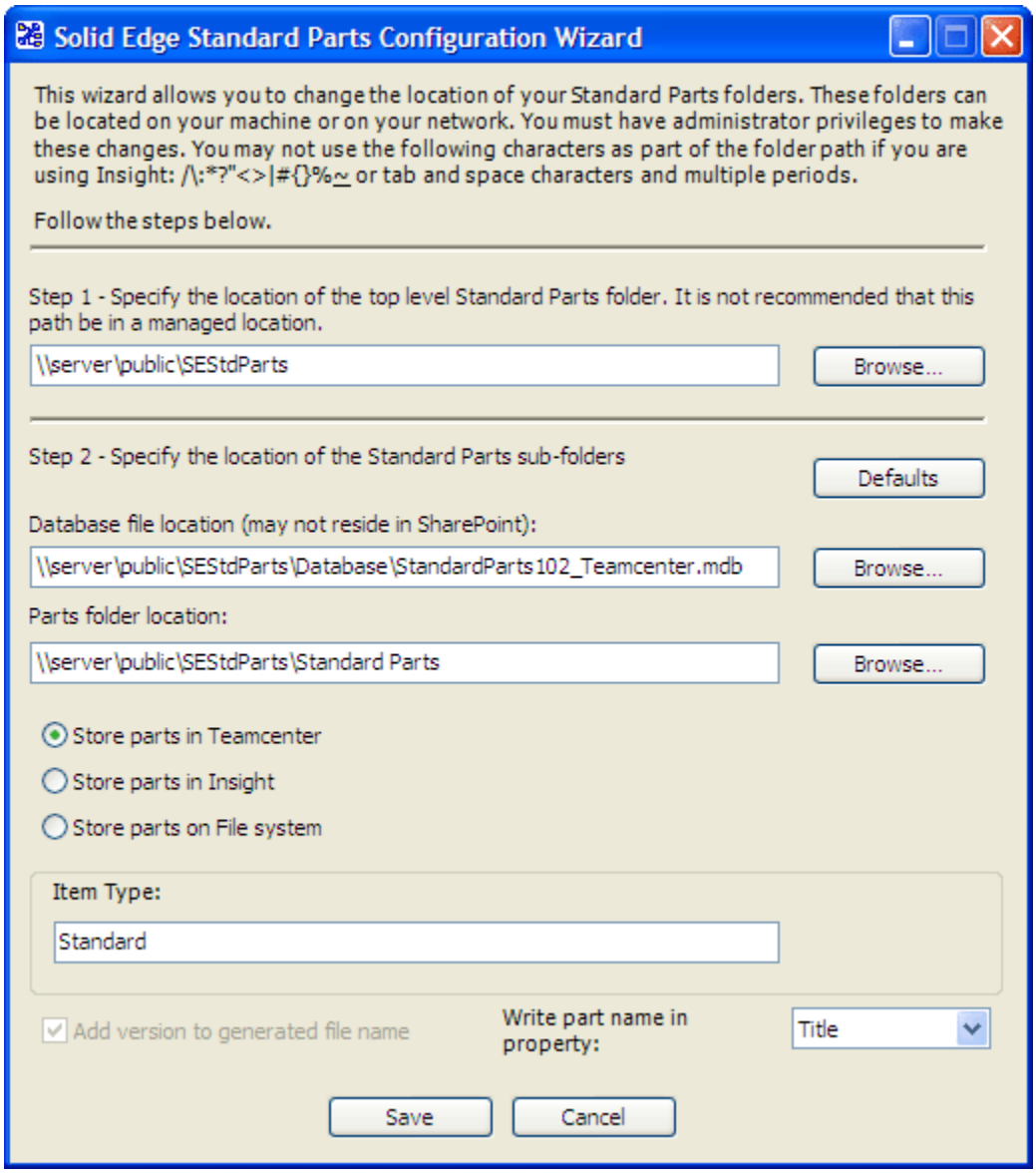

Solid Edge Standard Parts Administrator - InstallShield Wizard
Custom Setup
Select the program features you want installed.
Click on an icon in the list below to change how a feature is installed.
Master Part Files
Feature Description
This feature installs the Solid Edge Master Part Files
Solid Edge Standard Parts Administrator - InstallShield Wizard
Change Current Destination Folder
Click Next to accept the folder or choose your own folder.
Look in:
Solid Edge Standard Parts
Folder name:
\\Server\Public\SEStdParts
InstallShield
OK
Cancel

Standard Parts administration installation (Step 2).

3. Complete the installation.

[1] Full Solid Edge software must be installed. Solid Edge Insight Server or Insight Connect installation is not supported!!!

4. If you need to install the full version of the Standard Parts libraries, you can do it using the same method described in the section "Installing Libraries" (but you do not need to register your parts into database).
5. Launch the Configuration Wizard
    a. Select **Store parts in Teamcenter**.
    b. Set a new database.
    c. Click on "Save" to confirm the settings.


Solid Edge Standard Parts Configuration Wizard
This wizard allows you to change the location of your Standard Parts folders. These folders can be located on your machine or on your network. You must have administrator privileges to make these changes. You may not use the following characters as part of the folder path if you are using Insight: /\:*?"<>|#{}%~ or tab and space characters and multiple periods.
Follow the steps below.
Step 1 - Specify the location of the top level Standard Parts folder. It is not recommended that this path be in a managed location.
\\server\public\SEStdParts
Browse...
Step 2 - Specify the location of the Standard Parts sub-folders
Defaults
Database file location (may not reside in SharePoint):
\\server\public\SEStdParts\Database\StandardParts102_Teamcenter.mdb
Browse...
Parts folder location:
\\server\public\SEStdParts\Standard Parts
Browse...
Store parts in Teamcenter
Store parts in Insight
Store parts on File system
Item Type:
Standard
Add version to generated file name
Write part name in property:
Title
Save
Cancel

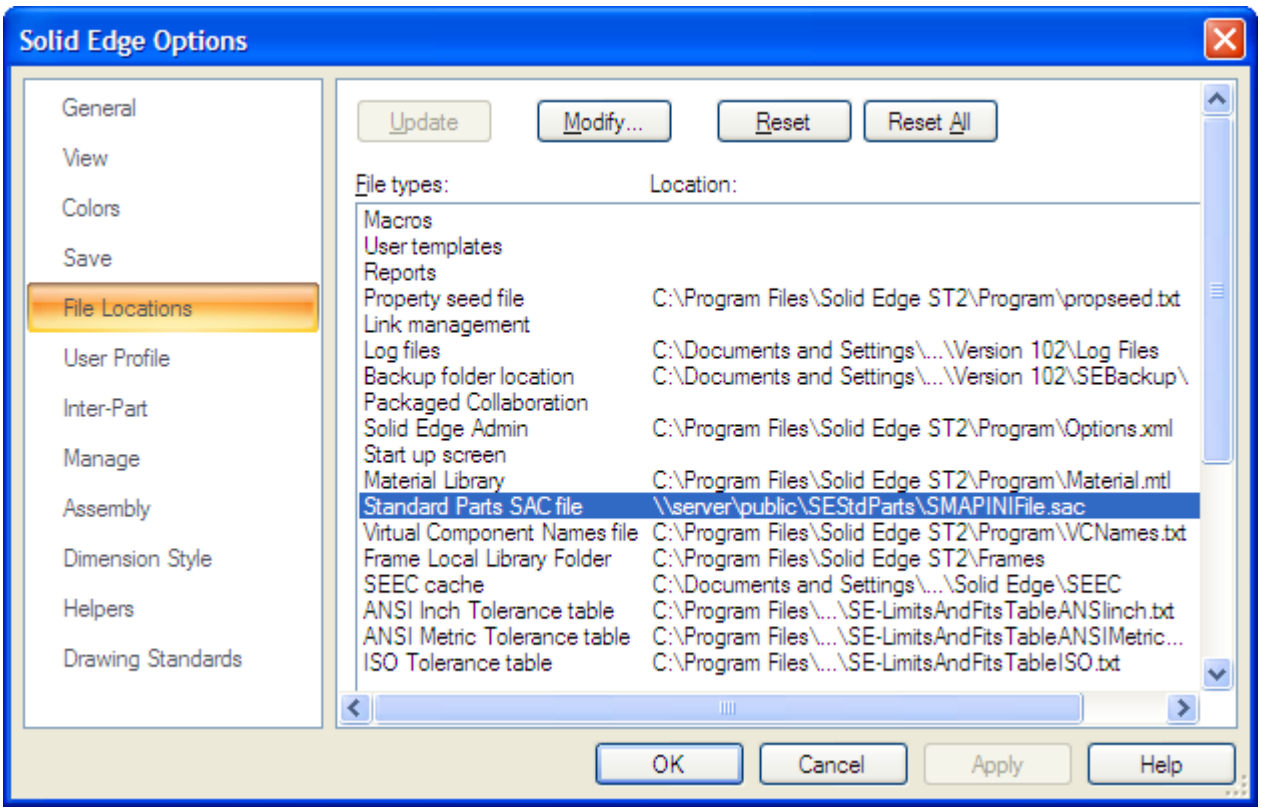
Standard Parts settings in Configuration Wizard

6. Launch the Solid Edge Standard Parts Administrator and add required standard parts into the database.

Standard Parts Workstation Installation (Teamcenter)

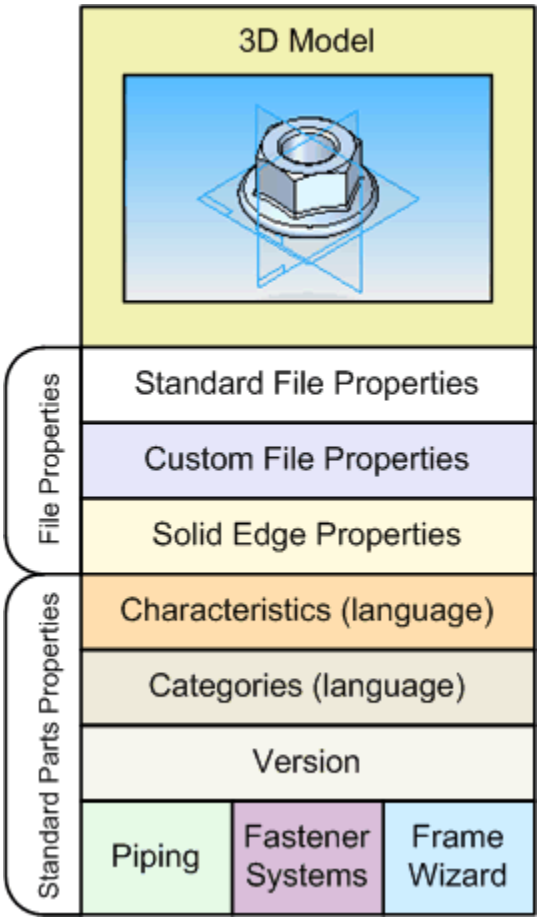
If the administration workstation is set up; on other workstations just set the Standard Parts SAC file location (Tools – Options – File Locations), the same as for a local or network installation. The SAC file must be located in the same UNC folder that was set during the Standard Parts administration installation.

Note: Teamcenter client and Solid Edge Embedded Client must be installed on all workstations.



Standard Part Files Definition

A Standard part in Solid Edge Standard Parts can be any Solid Edge native file (assembly, part, sheet metal, draft and weldment). An advantage of all standard parts is an additional information stream; a special property set called "Standard Parts Stream" (SMAP Stream). This stream is always included in the file. When the file is registered in the database, then the information from stream is also contained there. All Standard Parts Stream information can be changed or managed via the PartEditor.



Parent Part

The Parent Part is a global definition file. It contains a global 3D model and definition of all part variants (parent part filename suffix is "_Var_Gen.par"). A parent part is a global definition for a kind of standard (e.g. "*Bearing_DIN_ISO_3547_Bush_A_Var_Gen.par*"). When this file is registered by Solid Edge Standard Parts Administrator, all included single parts are registered in the database as virtual files.
Note: Parent parts are contained in your purchased Standard Parts Library.

Single Part

The Single Part file represents a single variant of a component included in a Parent part. A Single part can be included in your assembly (filename suffix *_vX.XX.par" where vX.XX is the version of file; e.g. "*Bush_ISO_3547_4_A_5.5x4_v9.00.par*").

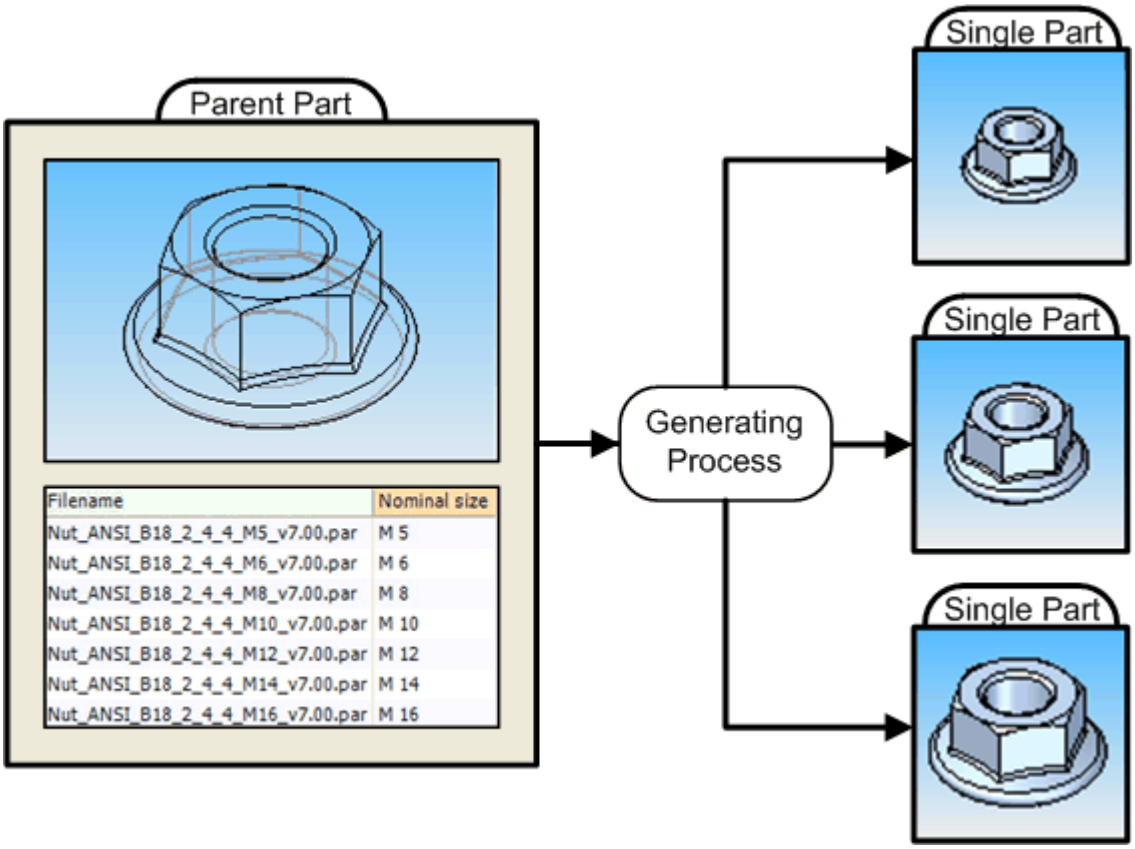
Non-generated Single Part

The Non-generated Single Part is a variant included in a Parent part which does not exist as an individual file. It is registered in the database (with 'Not generated' status). If a non-generated single part is needed in Solid Edge (e.g. it is to be placed from Standard Parts or selected by a Piping command to build a piping path) it must be generated first and then included in the assembly as a generated file. Single parts are generated automatically when they are placed for the first time. They can also be generated manually using Solid Edge Standard Parts Administrator.

Generated Single Part

Sometimes a single part is called a "generated part" or "generated variant". This file is created ("generated") from appropriate Parent part. This is the final file that is linked in your assembly.[1]

[1] Customer standard parts are managed in a similar way as generated single parts.

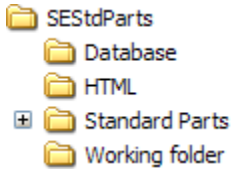

Parent Part
Filename
Nominal size
Nut_ANSI_B18_2_4_4_M5_v7.00.par
M 5
Nut_ANSI_B18_2_4_4_M6_v7.00.par
M 6
Nut_ANSI_B18_2_4_4_M8_v7.00.par
M 8
Nut_ANSI_B18_2_4_4_M10_v7.00.par
M 10
Nut_ANSI_B18_2_4_4_M12_v7.00.par
M 12
Nut_ANSI_B18_2_4_4_M14_v7.00.par
M 14
Nut_ANSI_B18_2_4_4_M16_v7.00.par
M 16
Generating Process
Single Part
Single Part
Single Part

Standard Parts Folder Structure and File Locations

The Standard Parts system uses various files. For correct operation all these files must be found in specific locations.
For the correct functionality the user should have at least "read" and "write" access permissions for the folders specified below in this chapter. To save the settings in the Solid Edge Standard Parts Configuration Wizard the user must also have administrator privileges.

Top Level Standard Parts Folder

The top level standard parts folder is the folder where "SMAPINIFile.sac" is saved. This file contains global Standard Parts settings and can be changed only using Solid Edge Standard Parts Configuration Wizard.
The default folder structure is shown to the right.[1]

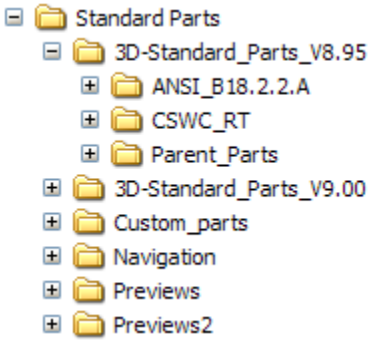


Database File Location

The database file location is where the Standard Parts database is stored. Except of a single user installation, the database must be always stored on a network drive.
Default database file location is the "Database" folder which is a sub-folder of the top level Standard Parts folder.

Note: Solid Edge Standard Parts Configuration Wizard and Database Manager distinguish the platform (file system, Insight, Teamcenter) on which the data written in the database reside (if the database contains some Standard Parts data). If the original and current platform differs, you will be called on to set another working database.

Parts Folder Location



The parts folder location is where the Standard Parts files are stored. The structure is shown to the right.
"3D-Standard_Parts_V8.95" and "3D-Standard_Parts_V9.00" are sub-folders with standard parts version 8.95 (Free Content) and 9.00.
Parent parts files are stored in a sub-folder "Parent_Parts".
The "Custom_parts" folder is a location where BulkPartAdder creates and stores customer standard parts. This folder is created automatically after the first customer standard part is created.
"Navigation" folder stores information on visual navigation which can be used in the Standard Parts application as one of the ways to find a required part.
"Previews" and "Previews2" folders contain supplementary GIF (Previews) and DXF (Previews2) previews, which serve mainly to clarify the meaning of particular characteristics of a part selected in the Standard Parts application.

Note: Standard Parts version 8.45 and above are being delivered with DXF previews only.
Note: When using Teamcenter, Standard Parts are generated to the folder "Teamcenter" (sub-folder of the top level Standard parts folder).

[1] Default definition is different for URL (Insight) location because of path/filename lengths and character limitations

Generated Parts Location

If a part is being used (opened or placed) for the first time, it must be generated. In this case a new single part file is created. This file is stored in a location which depends on the platform used – file system, Insight or Teamcenter.
If the file system is being used, single parts are generated in folders which are on the same level as the "Parent_Parts" folder. The folder names correspond with the end categories to which these single parts belong.
For an Insight installation, you can set a generated parts folder yourself. Single parts will be generated into its sub-folders corresponding with end categories as for the file system.
Teamcenter manages generated parts automatically (if it is set as the generated parts location).

Working Folder Location

Standard parts are normally generated into the locations described above. These paths are stored as links in the assemblies where the parts are placed. By activating the checkbox "Copy to working folder" (in the Standard Parts application) this default setting can be changed. In this case, the part is generated into its default folder and a copy of the part is created in the Working folder. This copy is then referenced in the assembly. This option is useful if you know you want to make an individual user part from some standard part (e.g. to bore a hole into a bolt). For parts with variable characteristics (such as profiles) the part is always copied into the Working folder because every part is unique (e.g. beams with different lengths created using one profile). The location of the Working folder can be either set to the default (subfolder of the Solid Edge Standard Parts folder) or set as the folder where the active assembly resides. The setting can be changed in the Standard Parts dialog (Tools – Options – checkbox "Use assembly folder as working folder"). A general definition of this folder is: the folder where the parts are copied to when the option "Copy to working folder" is set in Standard Parts dialog (if the "Use assembly folder as working folder" option is not set).

Note: Variable parts (parts with at least one variable characteristic - structural shapes and tubes typically) are always copied to working folder.

HTLM Folder Location

The HTML folder is a folder containing additional files associated with end categories. These files can be displayed using a special function from the Standard Parts application. HTML folder is created automatically as a subfolder of the top level Standard Parts folder.

Database

The Standard Parts document management system uses a Microsoft Access database. Users can manage the Solid Edge Standard Parts system via the Solid Edge Standard Parts Administrator application; this application can Add – Update – Register New – Delete CAD Standard Parts files within the database.

Document Version Schema

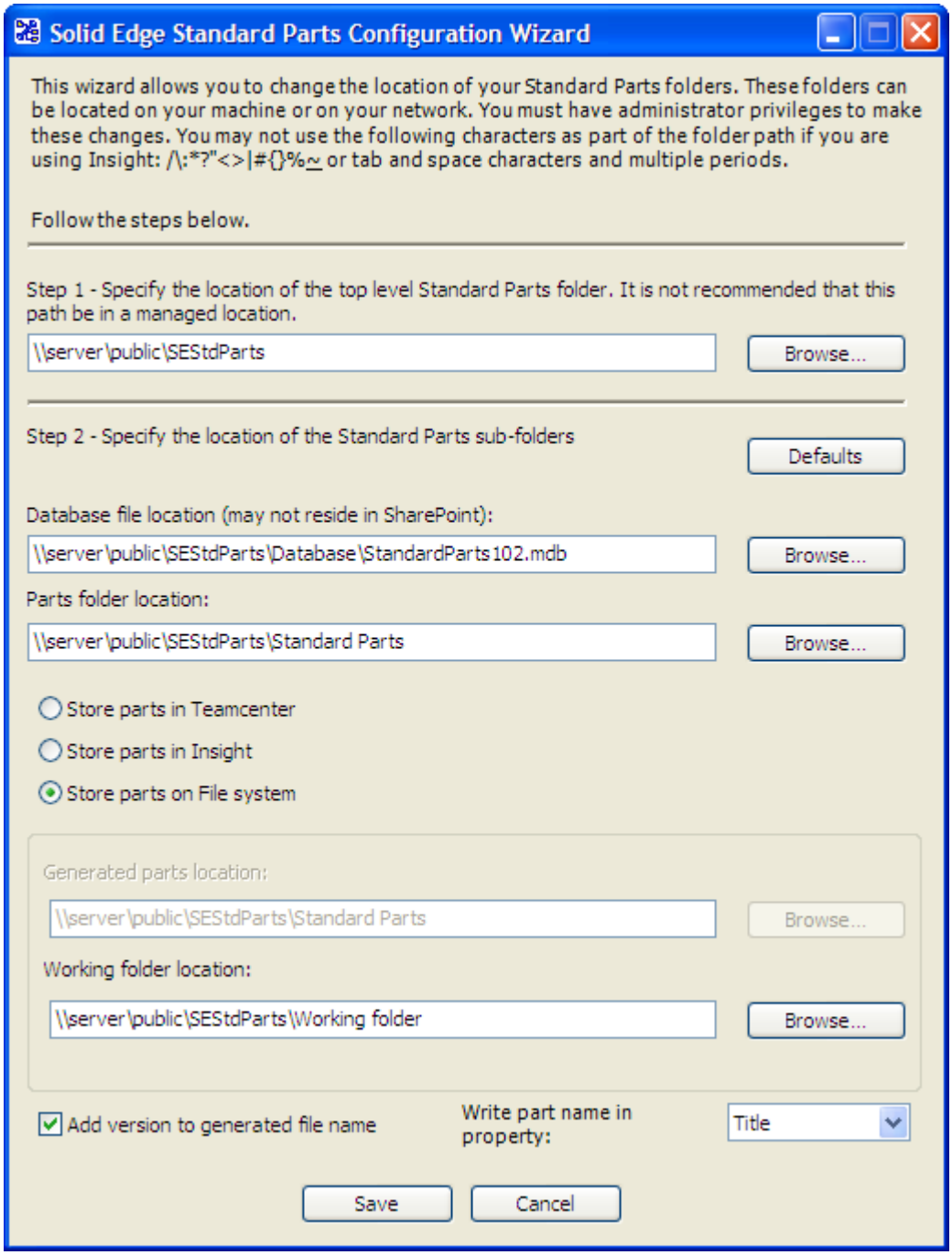
Standard Parts can be modified (e.g. norm or model changes). However, every standard part must be unique in the Standard Parts document management system. So, some parts must be included in the system (database) as the same part as an existing part, but with higher version. Solid Edge Standard Parts system can be managed with a Standard Part version. This means that all standard parts documents have their own information (property) about their version. The version is recognized and processed on every Solid Edge Standard Parts Administrator operation depending on the current settings. The most important usage is with the Add/Update Parts function: a user can easily keep his database up-to-date with the latest version of standards.

Solid Edge Standard Parts Configuration Wizard

The Solid Edge Standard Parts Configuration Wizard is a tool that helps you set the necessary paths (described in Standard Parts Folder Structure and File Locations section) needed for using Solid Edge Standard Parts. This tool allows creation of the entire directory structure needed for organization of the system and files. If you install Solid Edge Standard Parts on a local machine or network you do not need to launch this wizard (paths are set automatically), but there is a possibility to change settings. In the case of an Insight installation or just storing parts on Insight or Teamcenter, using this wizard is always necessary.

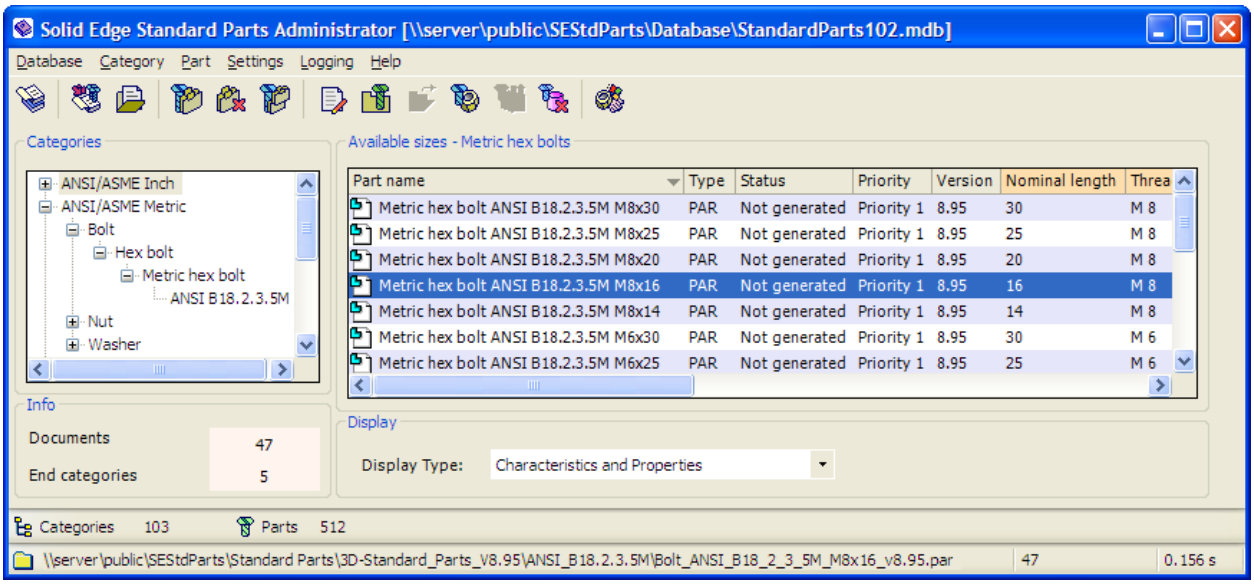
In "Step 1" the **top level Standard Parts folder** should be set. If you set this to an existing top level Standard Parts folder here, all settings in the Configuration Wizard will be set automatically according to the appropriate SAC file. Top level Standard Parts can be set to a file system folder or an Insight folder (Insight is not recommended but possible).
In "Step 2" the rest of necessary folders can be set. You can use the default values easily by clicking on "Defaults" button.
Database file location can only be set to a file system folder. The "Defaults" button sets the file name to "StandardParts102.mdb" in a "Database" folder, which is created as a sub-folder of the top level Standard Parts folder.
Default **parts** and **working folder locations** are "Standard Parts" and "Working folder" (sub-folders of the top level Standard Parts folder) if the "Store parts on file system" option is set. When using "Store parts in Insight" the parts and working folder must be an Insight location. Teamcenter manages these locations automatically (these cannot be modified) you can only set the Teamcenter **item type**. Generated parts and working folder must always be set on the same platform.
Single parts are usually generated with a version suffix in the file name (e.g. Pipe_DIN_2440_1x3.25_v9.00.par). If you require generation of single parts without this suffix (e.g. Pipe_DIN_2440_1x3.25.par), you can uncheck the "**Add version to generated file name**" box. This checkbox is available only when storing parts on the file system.
The **Write PartName in property** combo box allows the user to set a property where the Part name will be written during the generation process. Standard file properties (Author, Comments, Keywords, Subject, Title, Category, Company, Manager) and Solid Edge properties (Document Number, Revision Number, Project Name) can be used for this purpose. If "Store parts in Teamcenter" option is selected, Document Number and Revision Number are not available.

Note: When a standard part is being generated, the "Author" and "Company" properties are automatically filled with values from the system login. If the "Author" or "Company" property is set in the Solid Edge Standard Parts Configuration Wizard, it has a higher priority (a value from login will be overwritten with Part name).


Solid Edge Standard Parts Configuration Wizard
This wizard allows you to change the location of your Standard Parts folders. These folders can be located on your machine or on your network. You must have administrator privileges to make these changes. You may not use the following characters as part of the folder path if you are using Insight: /\:*?"<>|#{}%~ or tab and space characters and multiple periods.
Follow the steps below.
Step 1 - Specify the location of the top level Standard Parts folder. It is not recommended that this path be in a managed location.
\\server\public\SEStdParts
Browse...
Step 2 - Specify the location of the Standard Parts sub-folders
Defaults
Database file location (may not reside in SharePoint):
\\server\public\SEStdParts\Database\StandardParts102.mdb
Browse...
Parts folder location:
\\server\public\SEStdParts\Standard Parts
Browse...
Store parts in Teamcenter
Store parts in Insight
Store parts on File system
Generated parts location:
\\server\public\SEStdParts\Standard Parts
Browse...
Working folder location:
\\server\public\SEStdParts\Working folder
Browse...
Add version to generated file name
Write part name in property:
Title
Save
Cancel

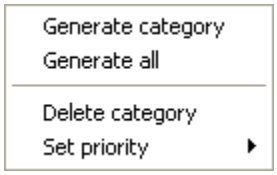
Solid Edge Standard Parts Administrator

The Solid Edge Standard Parts Administrator is an application designed for managing Standard Parts database/s. It allows you to create, delete and update a database, switch between databases, register parts into a database, generate parts, define part priorities[1], etc.


Solid Edge Standard Parts Administrator [\\server\public\SEStdParts\Database\StandardParts102.mdb]
Database Category Part Settings Logging Help
Categories
ANSI/ASME Inch
ANSI/ASME Metric
Bolt
Hex bolt
Metric hex bolt
ANSI B18.2.3.5M
Nut
Washer
Available sizes - Metric hex bolts
Part name Type Status Priority Version Nominal length Threa
Metric hex bolt ANSI B18.2.3.5M M8x30 PAR Not generated Priority 1 8.95 30 M 8
Metric hex bolt ANSI B18.2.3.5M M8x25 PAR Not generated Priority 1 8.95 25 M 8
Metric hex bolt ANSI B18.2.3.5M M8x20 PAR Not generated Priority 1 8.95 20 M 8
Metric hex bolt ANSI B18.2.3.5M M8x16 PAR Not generated Priority 1 8.95 16 M 8
Metric hex bolt ANSI B18.2.3.5M M8x14 PAR Not generated Priority 1 8.95 14 M 8
Metric hex bolt ANSI B18.2.3.5M M6x30 PAR Not generated Priority 1 8.95 30 M 6
Metric hex bolt ANSI B18.2.3.5M M6x25 PAR Not generated Priority 1 8.95 25 M 6
Info
Documents 47
End categories 5
Display
Display Type: Characteristics and Properties
Categories 103
Parts 512
\\server\public\SEStdParts\Standard Parts\3D-Standard_Parts_V8.95\ANSI_B18.2.3.5M\Bolt_ANSI_B18_2_3_5M_M8x16_v8.95.par
47
0.156 s

Solid Edge Standard Parts Administrator Main Form

The Solid Edge Standard Parts Administrator main form displays the categories and documents which are registered in the current working database.



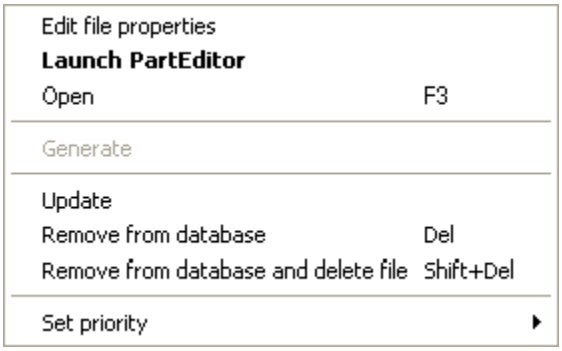
The "Categories" section shows a complete tree structure of all categories in the database. Using the context menu, you can perform various operations. You can generate parts in a selected category (including subcategories). You can also delete selected categories from the database (files are not removed from the hard drive) and set priority for a selected category. Priority can be used for filtering parts in the Standard Parts dialog where you can select the lowest priority that should be visible. In this case the "Use priority filter" option must be set (in menu Settings – Priorities).

There is also a possibility to generate all non-generated parts in the database at once. This operation may take a long time (depending on the number of non-generated parts).

In the "Available sizes" section there are parts (variants) displayed which are included in the selected end category. For each document, Part name (or Path), Type, Status, Priority and Version are always shown. You can choose from 4 types of views in the "Display" section. The "Characteristics and Properties" is the default view. If it is selected the characteristics (all parts included in the end category must have the same set of characteristics), standard file properties (Author, Comments, Keywords, Subject, Title, Category, Company, Manager) and Solid Edge properties (Material, Document Number, Revision Number, Project Name) are shown. You can also select the "Characteristics" or "Properties" view separately. Choosing "Selected properties" allows you to set your own type of view. In the "Selected properties" dialog (available via toolbar or Settings menu) you can select the properties to be displayed. All characteristics, standard file properties, Solid Edge properties, custom properties and virtual properties (Created,

[1] Priorities are only used with Customer Standard Parts and Siemens Library standard generated parts

Extension, Filename, Last accessed, Last modified, User) included in parts registered in the database are available.



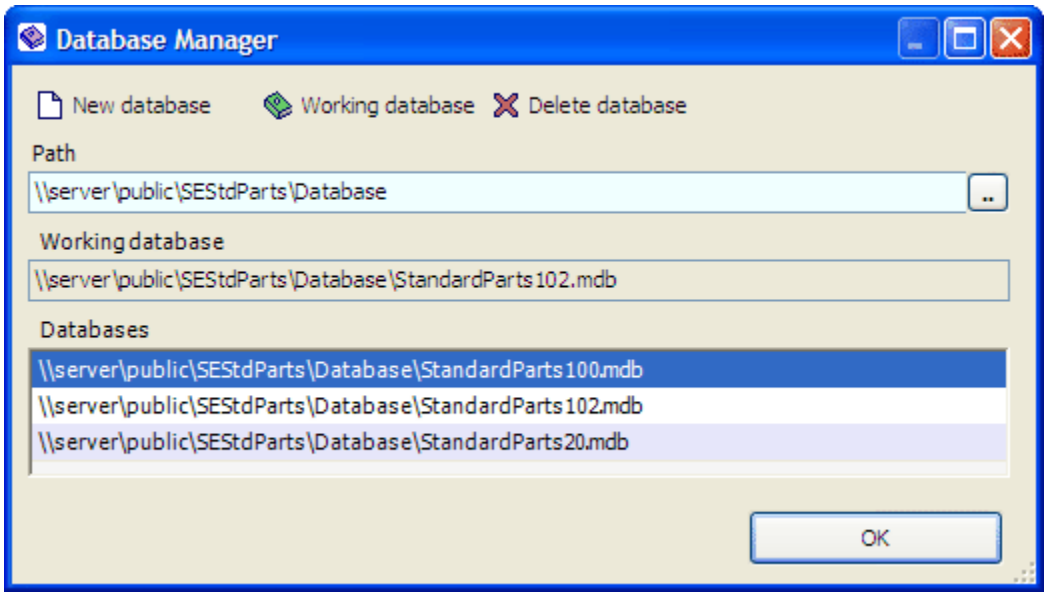
Using the context menu in the documents section you can perform some operations in addition to those that can be performed on the categories.
The "Edit file properties" option allows you to edit values of standard, Solid Edge and custom properties of selected document/s.
The "Launch PartEditor" command starts PartEditor. Using it you can change properties, characteristics, and categorization etc. of the selected document. This is recommended only for customer standard parts.
With the "open" command you can open selected parts in Solid Edge. It is active only for generated parts. You can also generate non-generated parts using the "Generate" command. The "Update" command updates the properties of the selected document (the properties are reread from the file).
You can also remove selected documents from the database and even delete the appropriate file (active only for generated documents).
There is also the possibility to set priority for each document individually.

In the "Info" section there are document and end category counts displayed for the currently selected category. In the row below the total number of categories and documents registered in the database is displayed.

In the bottom-most row of the Solid Edge Standard Parts Administrator form the full path of the selected document is displayed. This can also be displayed instead of "Part name" in the documents section. This can be set by checking "Show full path" in "Selected properties" dialog.

Database Manager

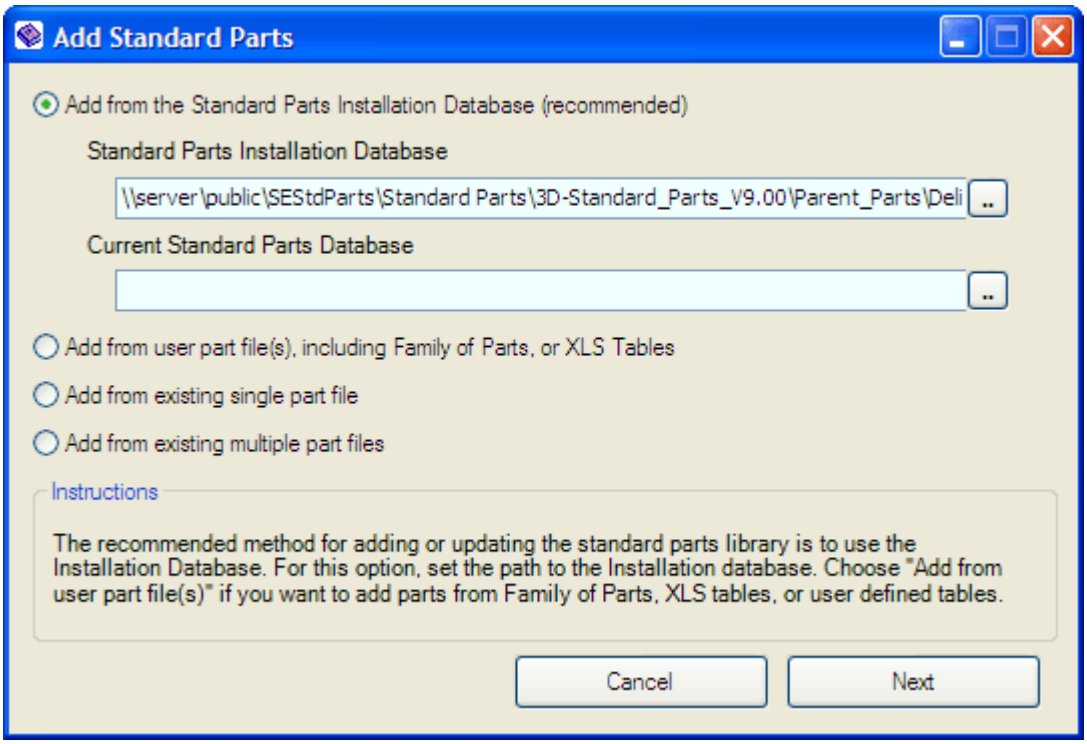
In the Database Manager you can create a database, set a database as a working database or delete a database. The working database cannot be deleted. If you want to delete it, you should set another database as the working database first.
In the "Databases" section there is a list of databases available in the location set in the "Path" box. The "Working database" (its file name with full path) is displayed in the box below the "Path" box.



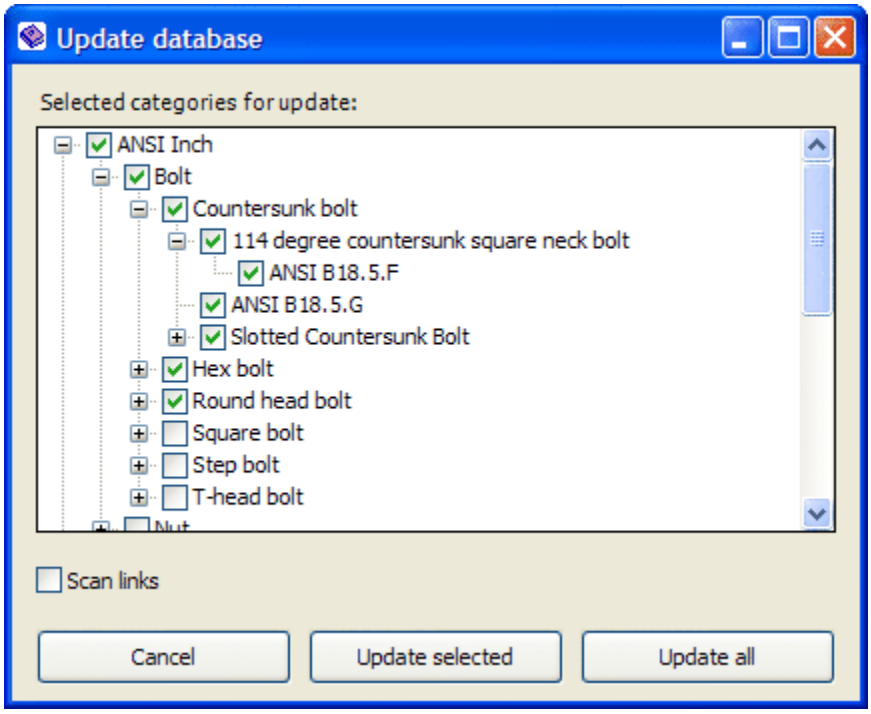
Add Parts Command

The most important function of the Solid Edge Standard Parts Administrator application is to register parts into the working database. After launching the Add Parts command the Add Standard Parts dialog box appears. It has four options to add parts from various locations and in various ways:

- **Add from the Standard Parts Installation Database** option uses the Installation Database for registering or updating standard parts in the database.
- **Add from user part file(s)** option serves for registering user documents as Standard Parts.
- **Add from existing single part file** allows registration of one single part or parent part in the database.
- **Add from existing multiple part files** option allows registration of multiple standard parts from specified location/s

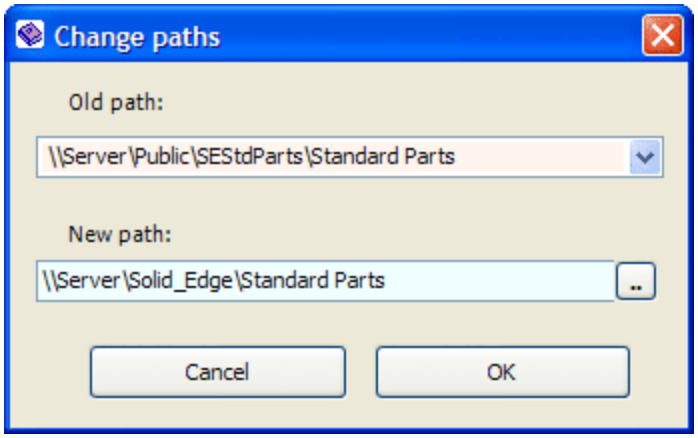

Add Standard Parts
Add from the Standard Parts Installation Database (recommended)
Standard Parts Installation Database
\\server\public\SEStdParts\Standard Parts\3D-Standard_Parts_V9.00\Parent_Parts\Deli
..
Current Standard Parts Database
..
Add from user part file(s), including Family of Parts, or XLS Tables
Add from existing single part file
Add from existing multiple part files
Instructions
The recommended method for adding or updating the standard parts library is to use the Installation Database. For this option, set the path to the Installation database. Choose "Add from user part file(s)" if you want to add parts from Family of Parts, XLS tables, or user defined tables.
Cancel
Next

Update Command



Updating the database is the process of rereading and updating information from previously registered parts in the database. During this operation no new documents are imported except when the 'Scan links' option is checked. In this case, Solid Edge Standard Parts Administrator scans the links of all documents (registered in database) and checks whether the linked documents are also registered in the database. If they are not, Solid Edge Standard Parts Administrator registers them with the database.
To update the whole database, use the "Update all" button. If you want to update only certain categories, check them in the list and use the "Update selected" button.

Change Path Command



The Change path command is useful when you need to migrate your documents to another location. The Change path command solves (changes) links at the database – not in the file! Migration in this case means moving e.g. your all Solid Edge Standard Parts Directory or Solid Edge Standard parts directory to other machine/disk, even if you are deciding to change your local system to a shared network system.
If, for example, the name of a server and therefore the path to the standard parts has changed, you can correct the path via the "Change paths" function. Usage is straightforward. Enter the old path (which you want to change). You can also select the path from list box. The path listed is the previously used path from the "Add parts" dialog. Then enter or browse the new path and click "OK" to start the process.
To check whether the path of the selected Standard parts has been changed correctly, click 'Edit properties' from the context menu. If it has not been changed correctly, you will receive an error message that the file was not found.

Note: The "Change paths" function works with text replacement, not with real file system paths (pay attention to the ending backslash). This function does not change any links to standard parts in Solid Edge assemblies! This function is not acceptable when you are migrating your system from UNC path to URL

path (Insight)! This function can change your database in an irregular way – this option is recommended only for administrators.

Compact Database Command

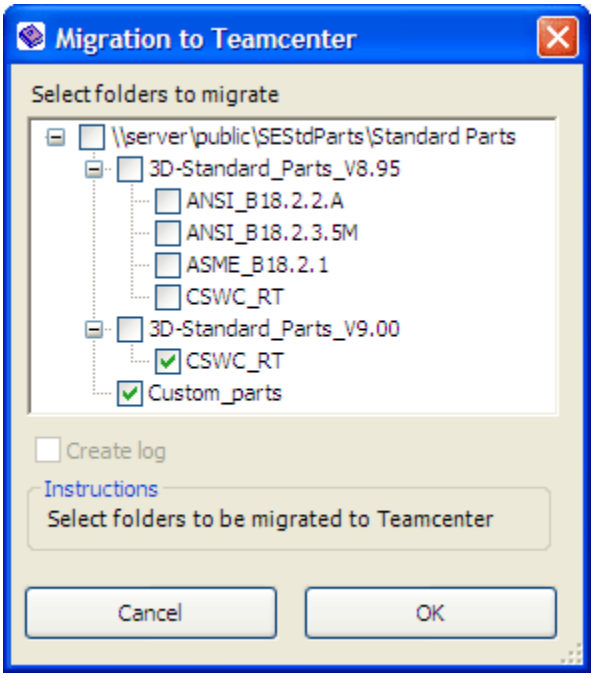
The Compact database command serves to reduce the database file size which grows automatically when operating with it. For the Microsoft Access database there is a recommendation that its size should be smaller than 1GB. When this limit is exceeded, the database may become unstable. The typical sign of the instability of the database is that available parts cannot be displayed (in the Solid Edge Standard Parts Administrator or Standard Parts application) for the selected end category.
The Compact database command allows you to reduce the database file size markedly in several seconds. Using this command you can avoid problems and increase Standard Parts performance.
We recommend the users to use the Compact database command in the following situations:

- after deleting a high number of categories/parts
- after updating the catalogs using SmartInstaller
- when the size of the database exceeds 1 GB
- when a problem occurs.

Note: Solid Edge Standard Parts Administrator is not necessary for database compression. It is also possible to launch the "MSAccessCompression.exe" application (located in the Solid Edge Program folder) with full path of the required database as a parameter.

E.g. MSAccessCompression.exe ||Server|Solid Edge Standard Parts|Database|StandardParts102.mdb

Migrate To Teamcenter Command

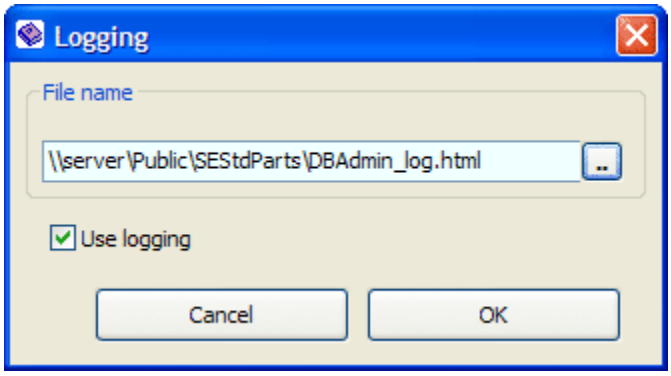


This function helps the users who have operated with Standard Parts on file system platform and want to switch to Teamcenter. The Migrate to Teamcenter command operates with generated parts only.
In the "Migration to Teamcenter" dialog you should select the Standard Parts folders whose content will be used with Teamcenter platform.
These folders will be then moved (including the folder structure) to the "Teamcenter" folder (sub-folder of the top level Standard parts folder). This is the only folder from which the generated parts can be added to the database (when Teamcenter platform is used).

Let us presume that there is a Standard Parts folder structure formerly used on file system. The typical (and recommended) migrating workflow is as follows:

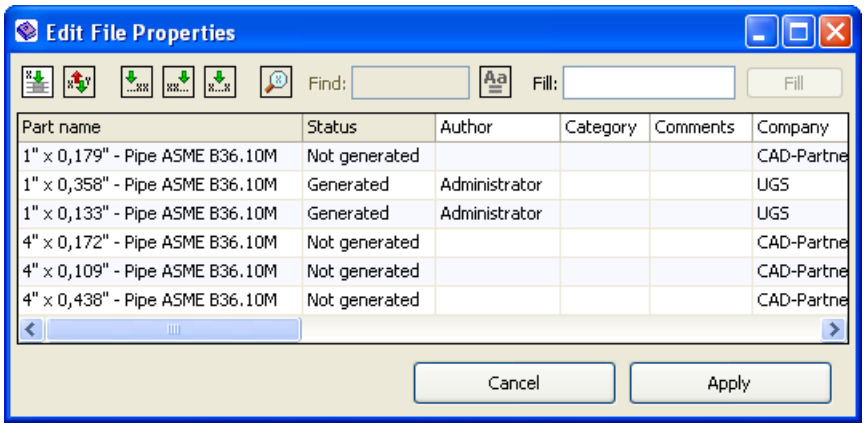
1. Launch Solid Edge Standard Parts Configuration Wizard. Browse to the desired top level Standard Parts folder (if needed). Then change the database name (to create a new one). Switch to "Store parts in Teamcenter" and set "Item Type" for Standard Parts (must exist in Teamcenter). Then click on "Save" to finish the Wizard (and save the changes to the SMAPINIFile.sac).
2. Launch Solid Edge Standard Parts Administrator. Start the "Migrate to Teamcenter" command from the Database menu. Then select (in the Migration to Teamcenter dialog) all the Standard Parts folders whose content you want to use with Teamcenter platform and confirm with "OK".
3. Use data prep utilities to analyze/modify generated parts before uploading to Teamcenter.
4. Use the "AddToTeamcenter" utility to add the parts to Teamcenter.
5. Add desired generated and also non-generated parts (parent parts) into the database.

Log File



The log file saves information about Solid Edge Standard Parts Administrator processes (registering parts and updating the database). The log file contains dates, times and descriptions of the processes, file names (with full paths) of the documents which were processed and the status of these processes (OK, Aborted, etc.).
With a click on the "Logging" button in the toolbar, the "Logging" dialog appears. Here you can set the log file (you can browse the location and enter file name). You can also enable or disable logging via the "Use logging" checkbox.

Edit File Properties



The "Edit File Properties" dialog box is a tool designed for mass editing of the values of standard parts file properties. It is not possible to create new custom properties there!
In the main grid the standard, Solid Edge and custom properties of selected parts are displayed. Each part can be identified by its Part name and Status (these values cannot be edited).
There are 6 functions represented with toolbar icons:

- Fill – fills up selected cells with a string set in the Fill field
- Replace – in selected cells replaces a string set in the Find field with a string set in the Fill field
- Insert prefix – sets a string typed in the Insert field as a prefix of strings in selected cells
- Insert suffix – sets a string typed in the Insert field as a suffix of strings in selected cells
- Insert – places a string set in the Insert field to a position specified in the Insert field
- Find – searches for and marks with bold font the cells which contain a string set in the find box. Using an additional icon you can set case-sensitive or insensitive searching.

Each operation is being launched via clicking the "action" button in the upper right corner of the "Edit File Properties" dialog. Its label (Fill, Replace, Insert or Find) depends on the selected function.
The "Apply" button in the bottom right corner applies all the changes made in the dialog box to the database and generated and non-generated part files. If you are going to change file properties of some read-only files (generated parts typically), the confirmative dialog will appear.

Note: Because the changes of property values are applied to part files (generated but also non-generated), the process (after clicking the Apply button) may take some time.

Creating Customer Standard Parts

In Standard Parts there is a special tool for creating customer standard parts. This tool is integrated in the Solid Edge Standard Parts Administrator. This wizard is called BulkPartAdder. You can launch it by selecting "Add new part file(s)" in the Add Purchased Standard Parts dialog (Add parts command) and then clicking on the "Next" button.
In the first step you can select one of three methods to add your custom parts (these are dependent on your source file):

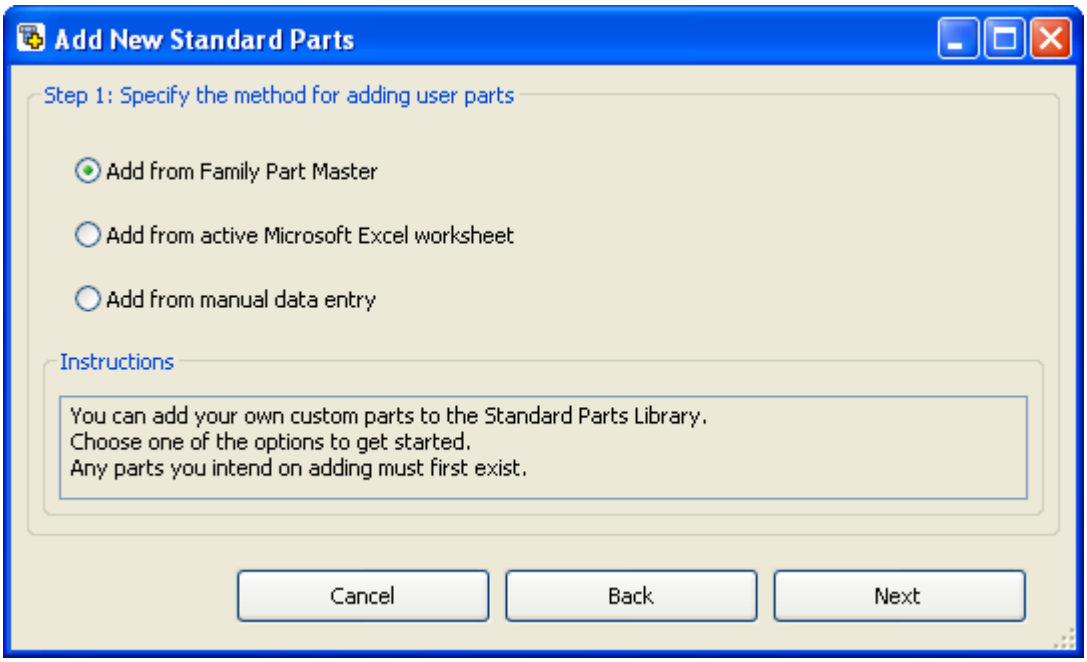
- Add from Family Part Master – you can add all the Family of Parts members at once.
- Add from active Microsoft Excel sheet – you can use an Excel table (either the whole table or just a part of it) which contains information on the parts you want to add.
- Add from manual data entry – you can manually select the part file(s) which you want to add to the database as standard part(s).

Creating Customer Standard Parts from Family Part Master

This method allows easy registration of an entire Family of Parts using its Part Master.

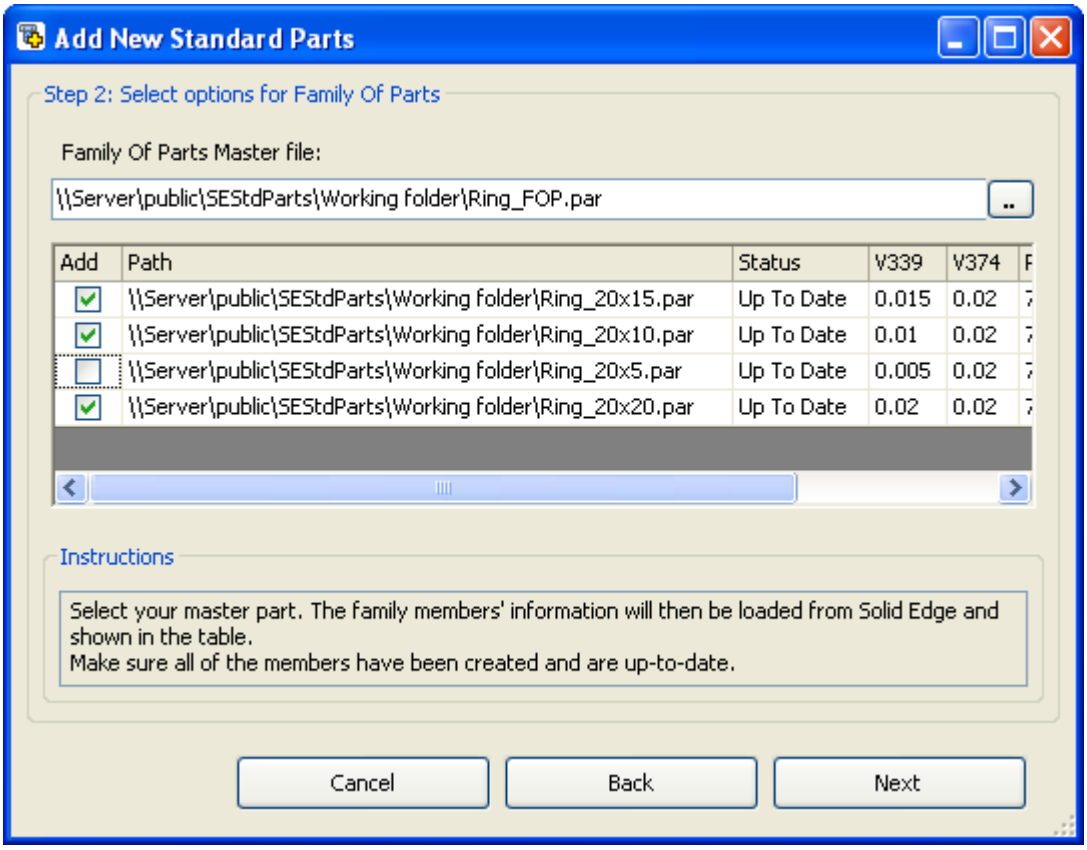
Step 1

Select "Add from Family Part Master" option and click on "Next".


Add New Standard Parts
Step 1: Specify the method for adding user parts
Add from Family Part Master
Add from active Microsoft Excel worksheet
Add from manual data entry
Instructions
You can add your own custom parts to the Standard Parts Library.
Choose one of the options to get started.
Any parts you intend on adding must first exist.
Cancel
Back
Next

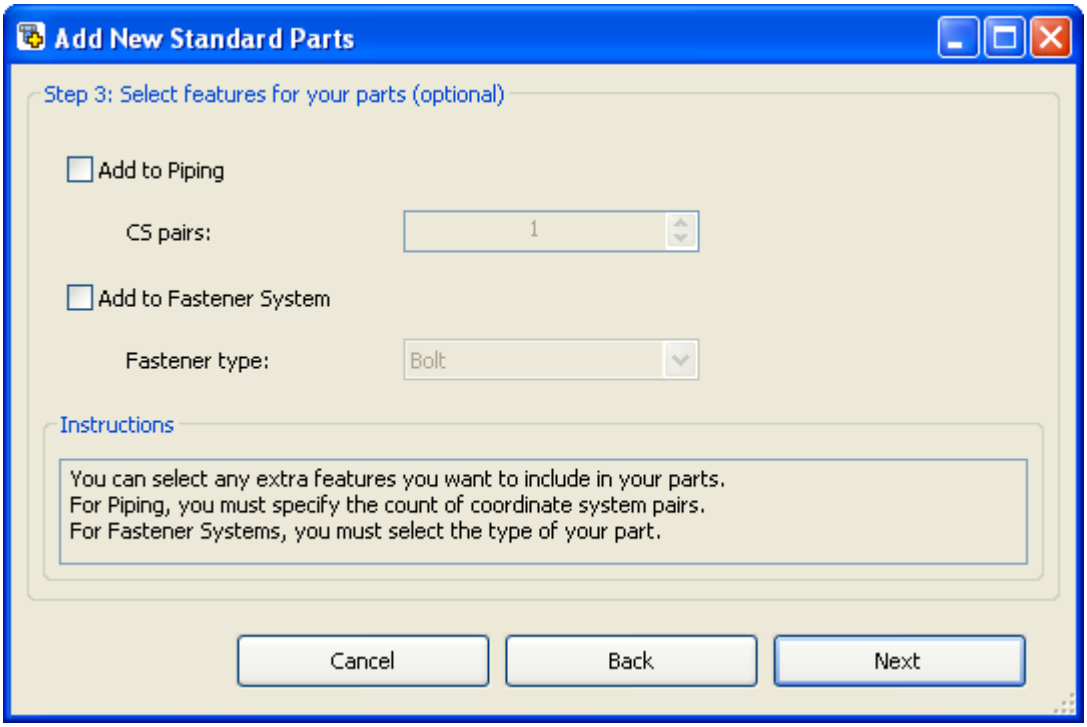
Step 2

Browse (via the browse button) to the required Family Master Part. Members of the family of parts will appear with full path, status and all variables which have values set in the Family of Parts. Family members must be created and up-to-date. In the "Add" column you can select the members which should be processed.



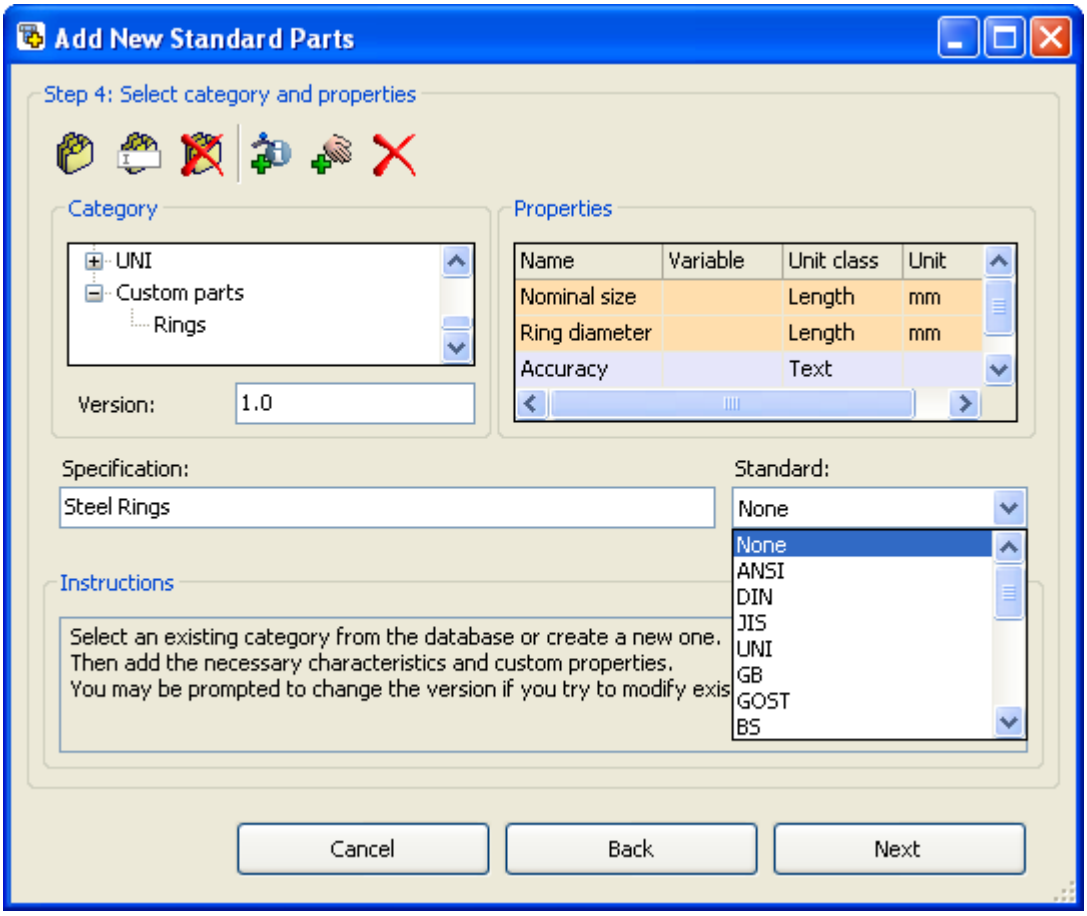
Step 3

Settings in this step are optional. If you want your custom part to be used as a fitting (or tube) by Solid Edge Piping you can check "Add to Piping" and set a number of coordinate systems pairs. To create a custom part to be used in Fastener Systems, check "Add to Fastener System" and select appropriate Fastener type (bolt, screw, washer or nut).


Add New Standard Parts
Step 3: Select features for your parts (optional)
Add to Piping
CS pairs:
1
Add to Fastener System
Fastener type:
Bolt
Instructions
You can select any extra features you want to include in your parts.
For Piping, you must specify the count of coordinate system pairs.
For Fastener Systems, you must select the type of your part.
Cancel
Back
Next

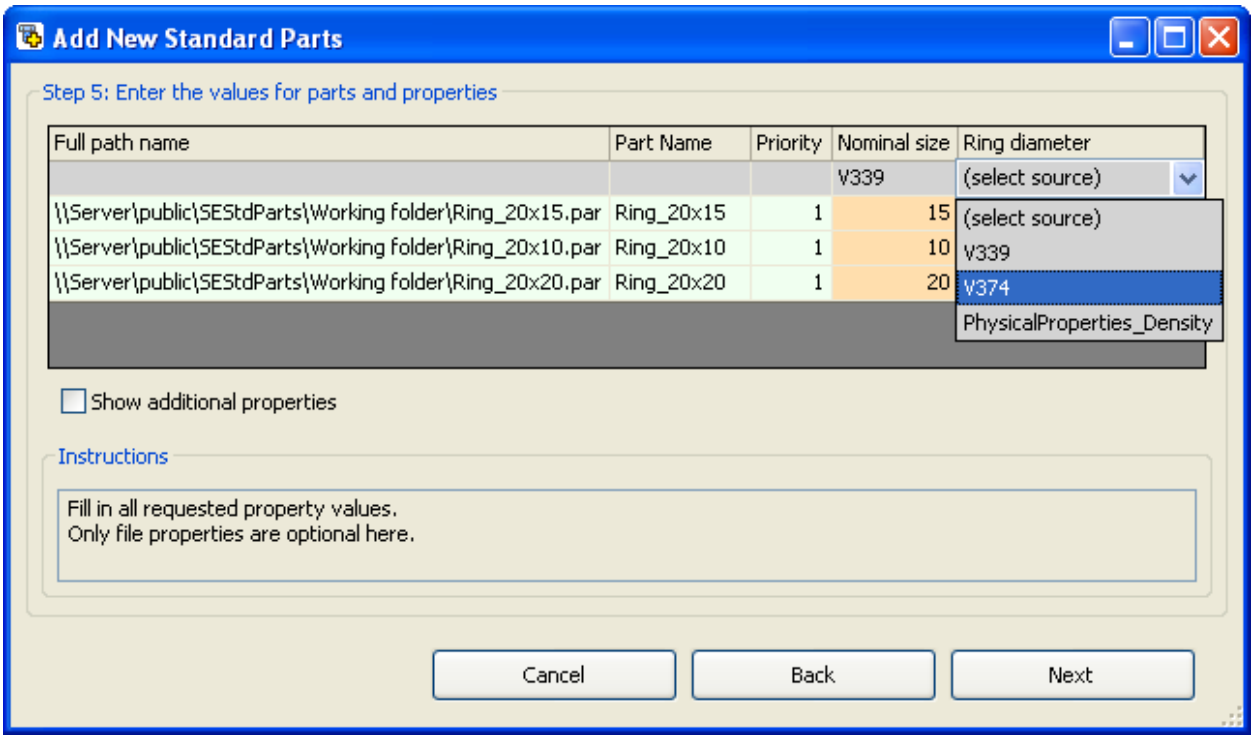
Step 4

In this step you should set a category for your custom part(s). Each category should have at least one characteristic assigned to it. If you want to add your custom parts to an existing category, you have to use the same characteristic set or change (typically increase) the category version. You can create two types of characteristics – constant or variable. For variable characteristics a variable must be set in the appropriate section of the Properties grid (this variable should correspond with an existing variable in the selected Family Master Part). For each characteristic you should set a unit class and the appropriate unit. In the Properties section there is also the possibility to add (or remove) custom properties. You can also enter a specification and set a standard for the created end category.



Step 5

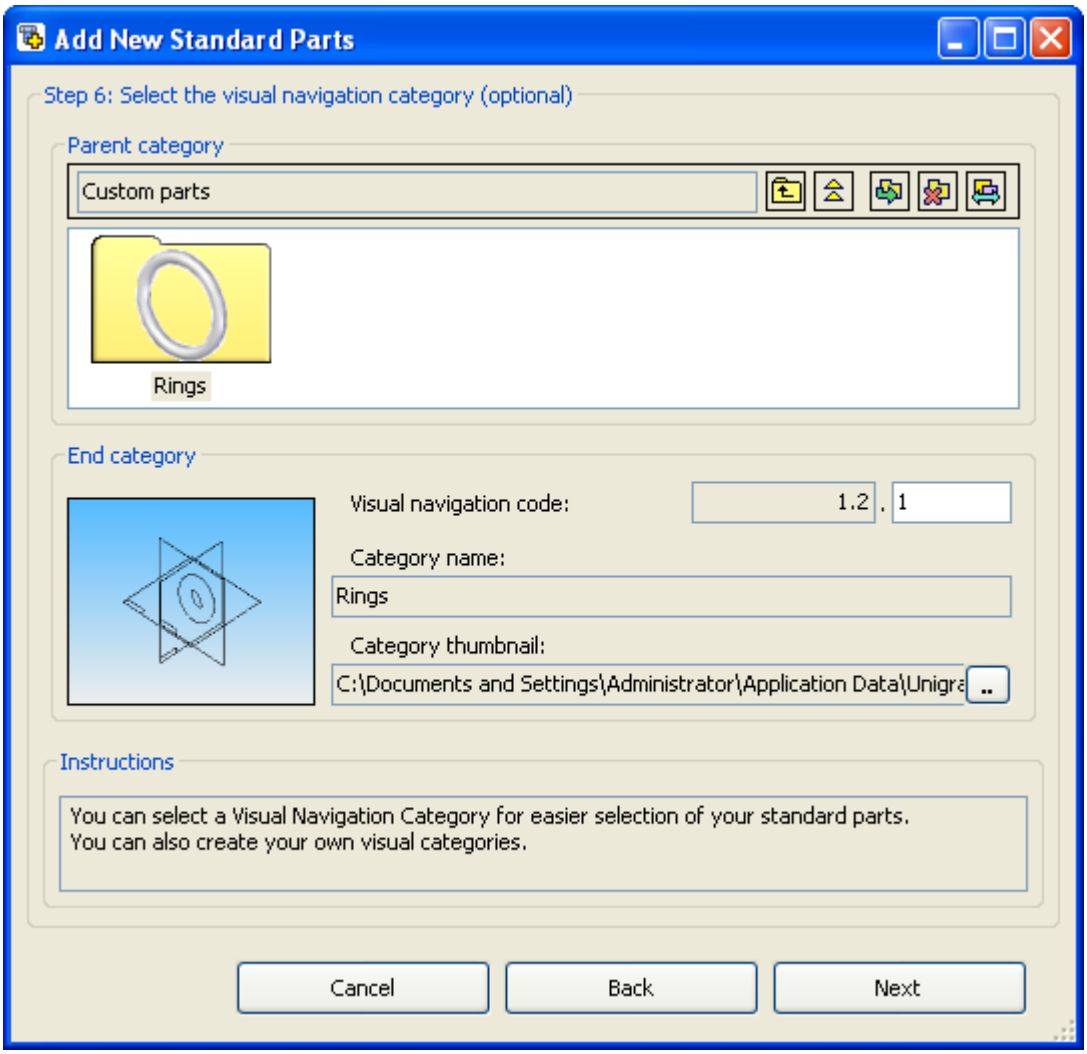
In step 5 the values of the characteristics (set in the previous step) should be set for each part (family member). If you click on the "(select source)" field (below the characteristic name) a list of variables (from the Family of Parts) will appear. Selecting one of them will cause its values to be filled out in the appropriate column. All values for all characteristics must be set.
If you check the "Show additional properties" box, then the file properties (standard, Solid Edge and custom) will be also displayed in the grid. Filling in these values is optional.
Using the context menu you can also remove a selected part row (at least one row must remain).



Step 6

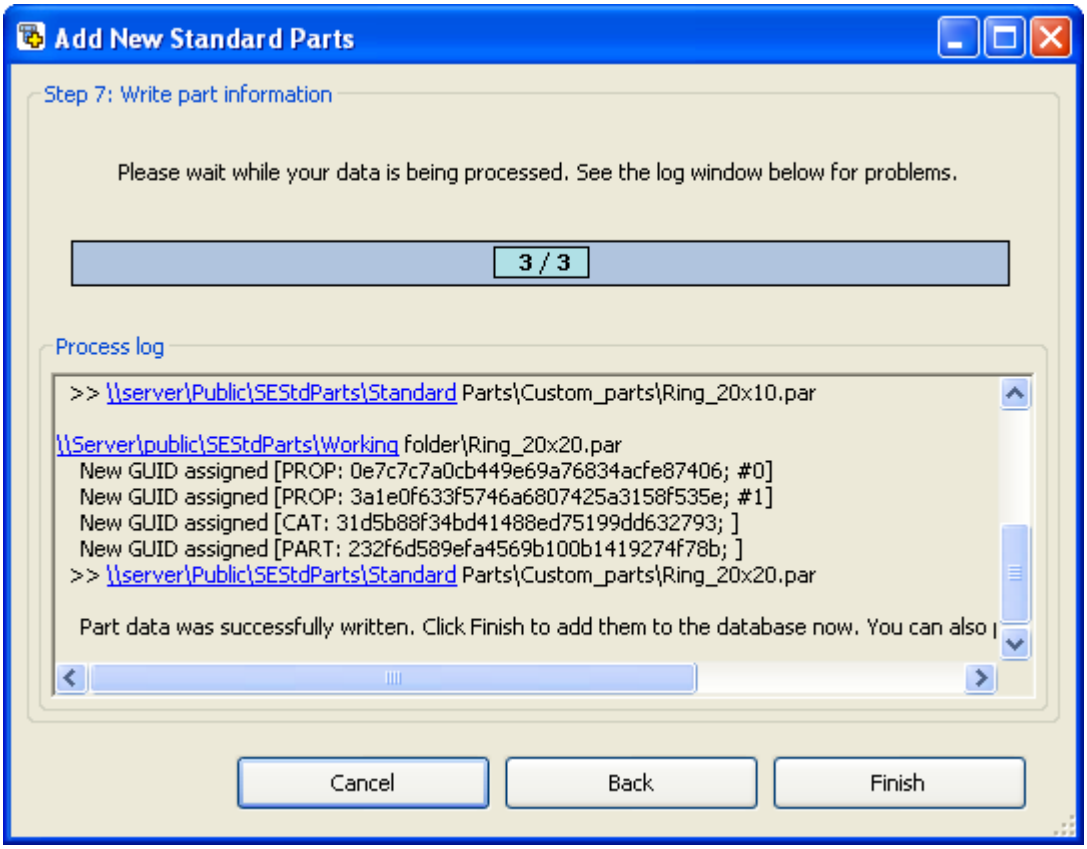
In this step you are required to create a Visual Navigation category for your newly created category. Creation of a Visual Navigation category structure requires the following steps:

1. Click on the "Add node" button to create new visual navigation node.
2. Confirm new visual navigation code.
3. Set a name for new node.
4. Click on the "Change thumbnail" button and select an image to replace the visual navigation code on the visual category's thumbnail.
5. Change the category thumbnail for the end category if necessary (preview of the part is selected by default).
6. Click on "OK" to confirm your newly created visual navigation category.



Step 7

In this step the necessary part information is created automatically at first. If you click on "Finish", the required parts will be copied to the Custom_parts folder (in the generated parts file location) and registered in the database.


Add New Standard Parts
Step 7: Write part information
Please wait while your data is being processed. See the log window below for problems.
3 / 3
Process log
>> \\server\Public\SEStdParts\Standard Parts\Custom_parts\Ring_20x10.par
\\Server\public\SEStdParts\Working folder\Ring_20x20.par
New GUID assigned [PROP: 0e7c7c7a0cb449e69a76834acfe87406; #0]
New GUID assigned [PROP: 3a1e0f633f5746a6807425a3158f535e; #1]
New GUID assigned [CAT: 31d5b88f34bd41488ed75199dd632793;]
New GUID assigned [PART: 232f6d589efa4569b100b1419274f78b;]
>> \\server\Public\SEStdParts\Standard Parts\Custom_parts\Ring_20x20.par
Part data was successfully written. Click Finish to add them to the database now. You can also
Cancel
Back
Finish

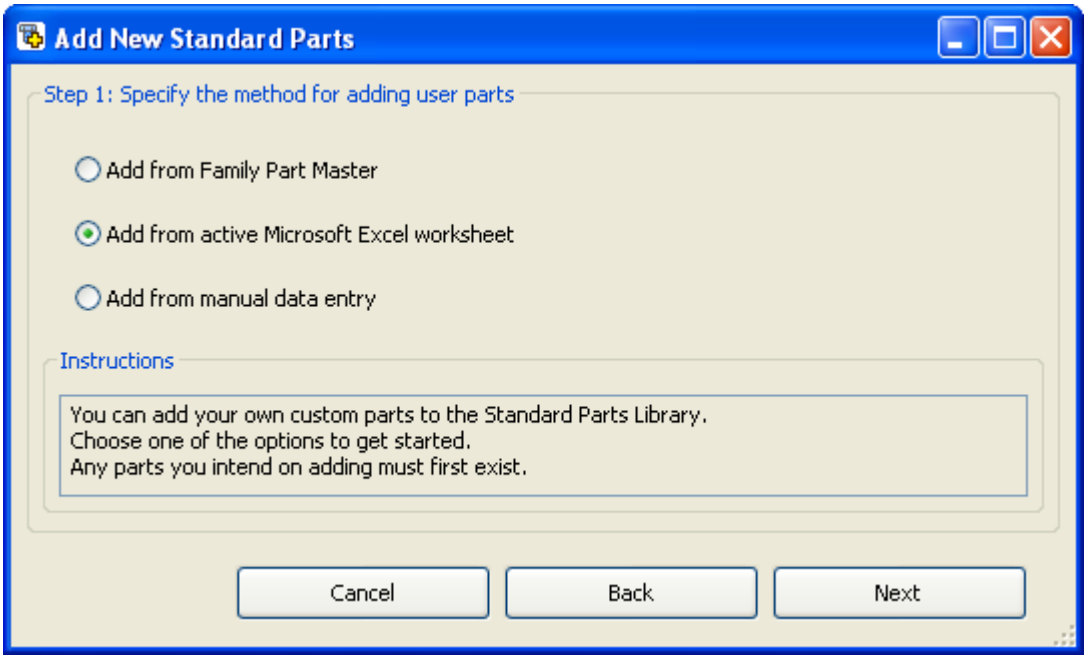
Creating Customer Standard Parts from Excel worksheet

This method may be suitable when you have created an Excel table which contains information about parts you want to register as customer standard parts (see example below).

	A	B	C	D
1	**Path**	**Diameter**	**Height**	**Part name**
2	\\server\Public\SEStdParts\Working folder\Cylinder20x40.par	20	40	Cylinder 20x40
3	\\server\Public\SEStdParts\Working folder\Cylinder30x40.par	30	40	Cylinder 30x40
4	\\server\Public\SEStdParts\Working folder\Cylinder30x50.par	30	50	Cylinder 30x50

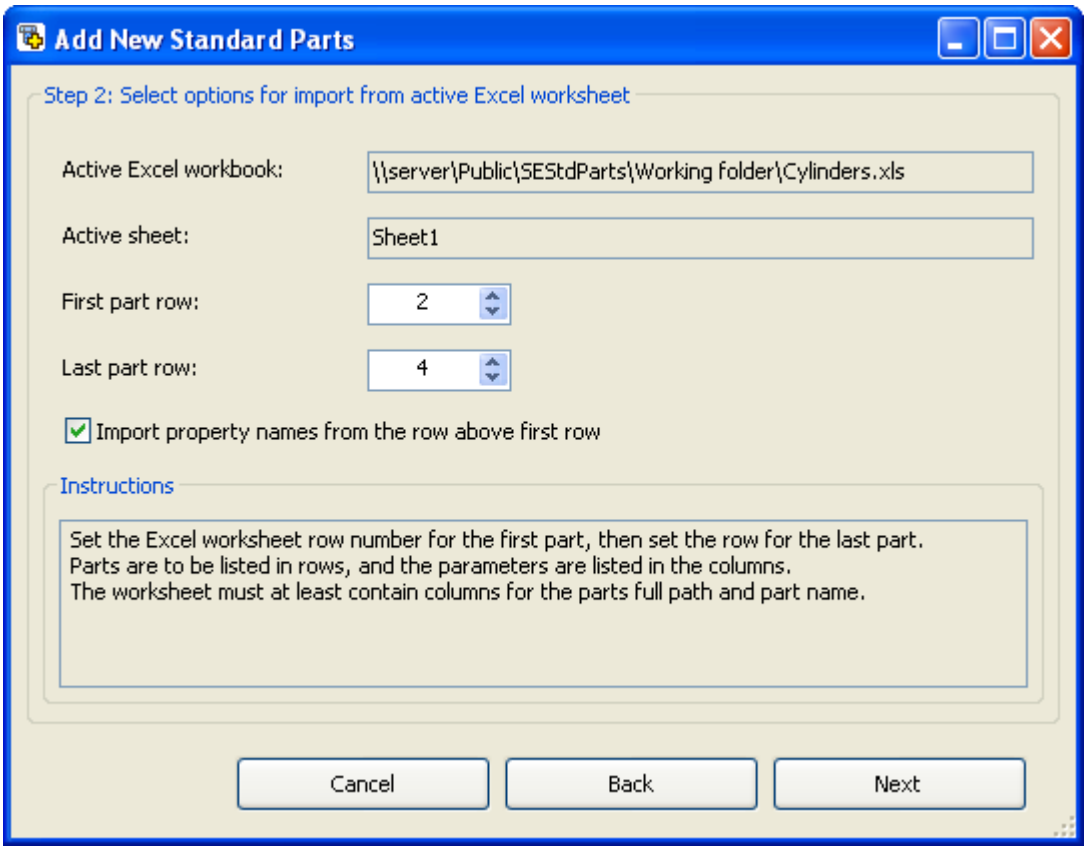
Step 1

Select "Add from active Microsoft Excel worksheet" option and click on "Next" (required Excel worksheet must be opened).


Add New Standard Parts
Step 1: Specify the method for adding user parts
Add from Family Part Master
Add from active Microsoft Excel worksheet
Add from manual data entry
Instructions
You can add your own custom parts to the Standard Parts Library.
Choose one of the options to get started.
Any parts you intend on adding must first exist.
Cancel
Back
Next

Step 2

In this step you should select the rows (from the active sheet) which will be processed. These are rows 2 to 4 (in our case). By checking a box you can also import names for each column in the sheet. It could serve for better identification of values in step 5.

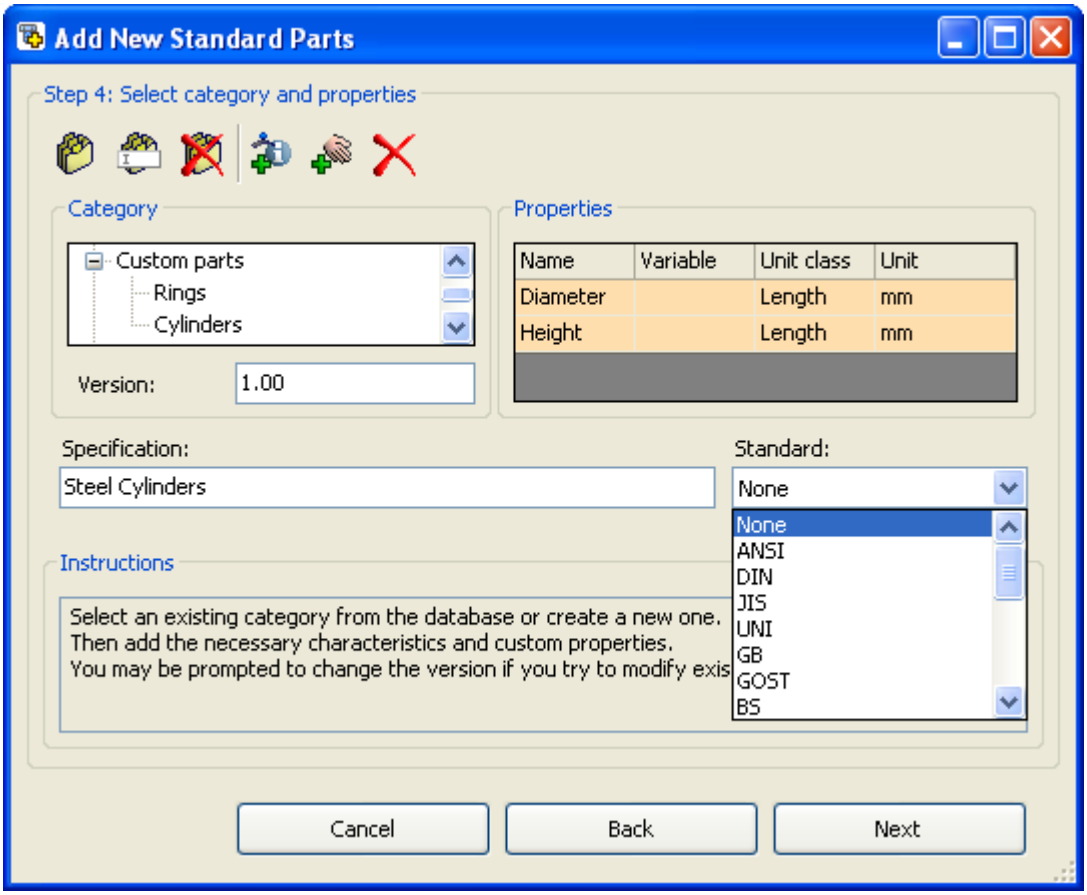

Add New Standard Parts
Step 2: Select options for import from active Excel worksheet
Active Excel workbook:
\\server\Public\SEStdParts\Working folder\Cylinders.xls
Active sheet:
Sheet1
First part row:
2
Last part row:
4
Import property names from the row above first row
Instructions
Set the Excel worksheet row number for the first part, then set the row for the last part.
Parts are to be listed in rows, and the parameters are listed in the columns.
The worksheet must at least contain columns for the parts full path and part name.
Cancel
Back
Next

Step 3

Settings in this step are optional. If you want your custom part to be used as a fitting (or tube) by Solid Edge Piping you can check "Add to Piping" and set a number of coordinate systems pairs. To create a custom part to be used in Fastener Systems, check "Add to Fastener System" and select appropriate Fastener type (bolt, screw, washer or nut).

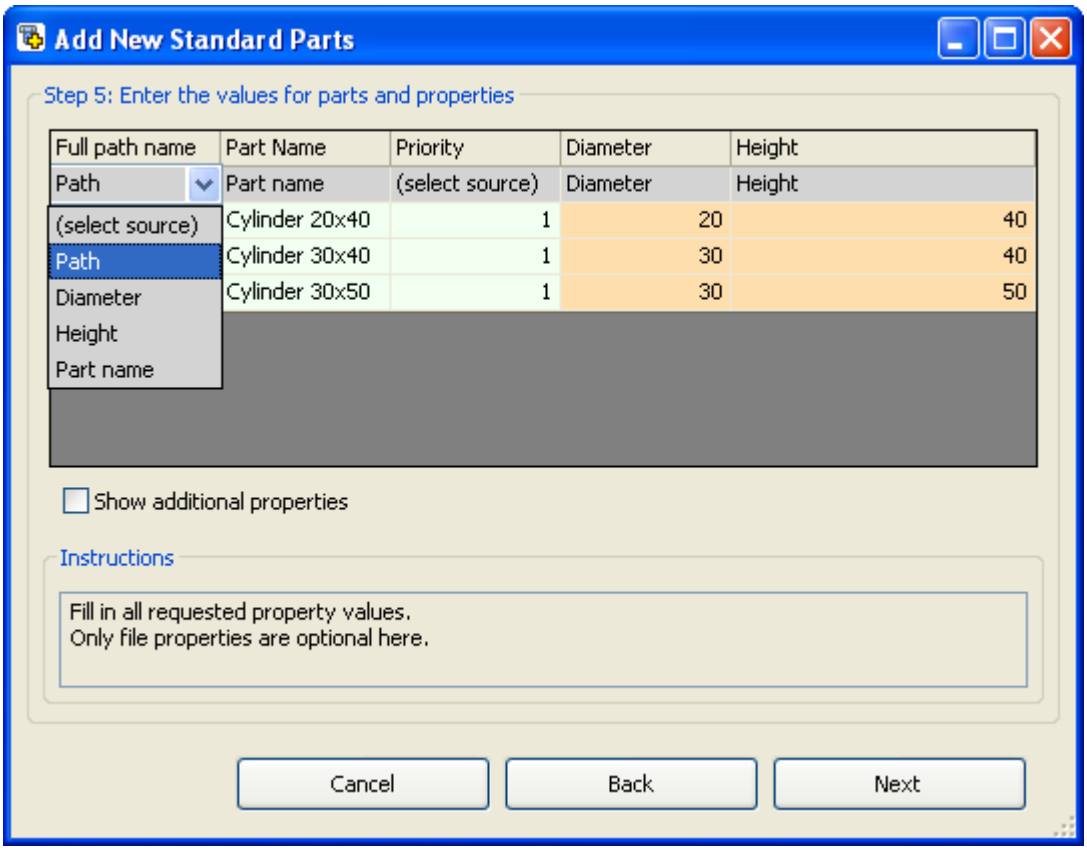
Step 4

In this step you should set a category for your custom part(s). Each category should have at least one characteristic assigned to it. If you want to add your custom parts to an existing category, you have to use the same characteristic set or change (typically increase) the category version. You can create two types of characteristics – constant or variable. For variable characteristics a variable must be set in the appropriate section of the Properties grid (this variable should correspond with an existing variable in the selected Family Master Part). For each characteristic you should set a unit class and the appropriate unit. In the Properties section there is also the possibility to add (or remove) custom properties. You can also enter a specification and set a standard for the created end category.



Step 5

This step requires, at a minimum, values for Full path, Part Name and characteristics set in previous step to be entered. For each column there is a drop down list "(select source)" containing the labels of columns in the active Excel worksheet. It allows mapping of values from the active Excel worksheet to the appropriate columns that are displayed in this step. You can also set (or edit) values manually.
If you check the "Show additional properties" box, then file properties (standard, Solid Edge and custom) will also be displayed in the grid. Filling out these values is optional.



Step 6

In this step it is possible to create a Visual Navigation category for your newly created category. This is the same as for creating customer standard parts from a family part master.

Note: If you are registering your custom parts into an existing end category, this step will be skipped automatically.

Step 7

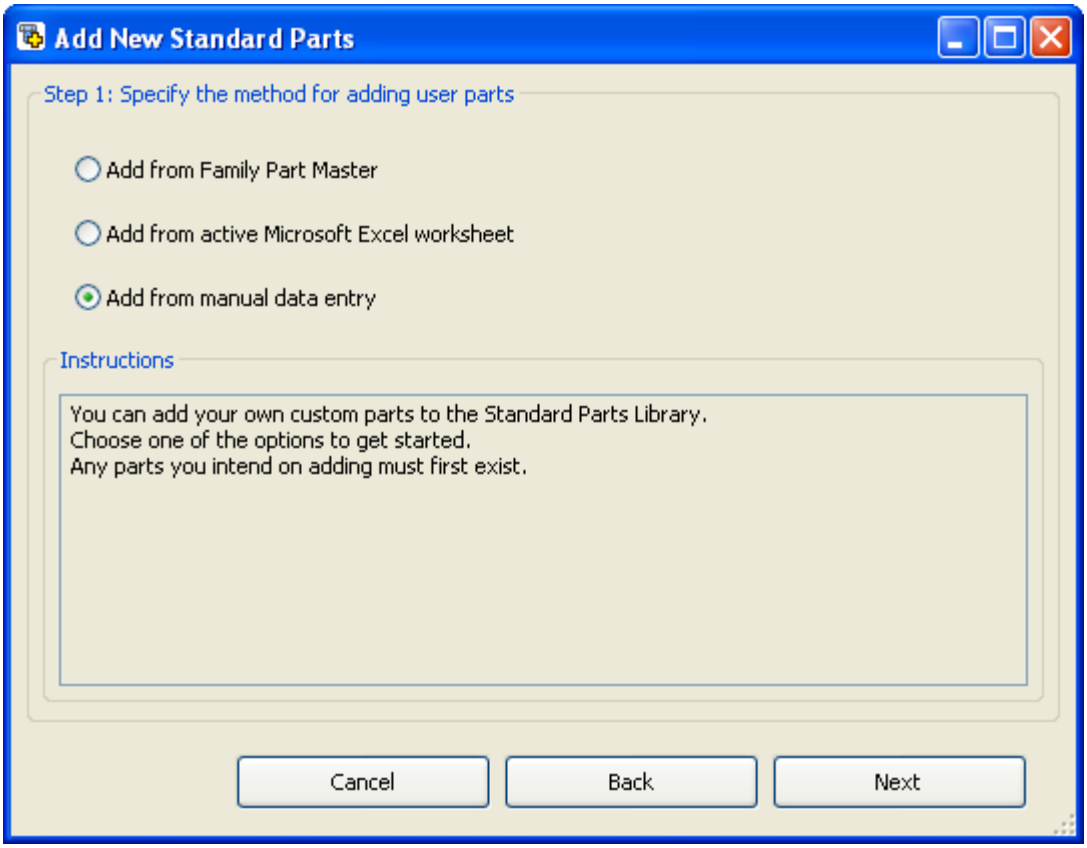
In this step the necessary part information is created automatically at first. If you click on "Finish", the required parts will be copied to the Custom_parts folder (in the generated parts file location) and registered in the database.

Creating Customer Standard Parts from Manual Data Entries

This is a method for common user parts. You can just manually select required parts (or part locations) and complete the wizard.

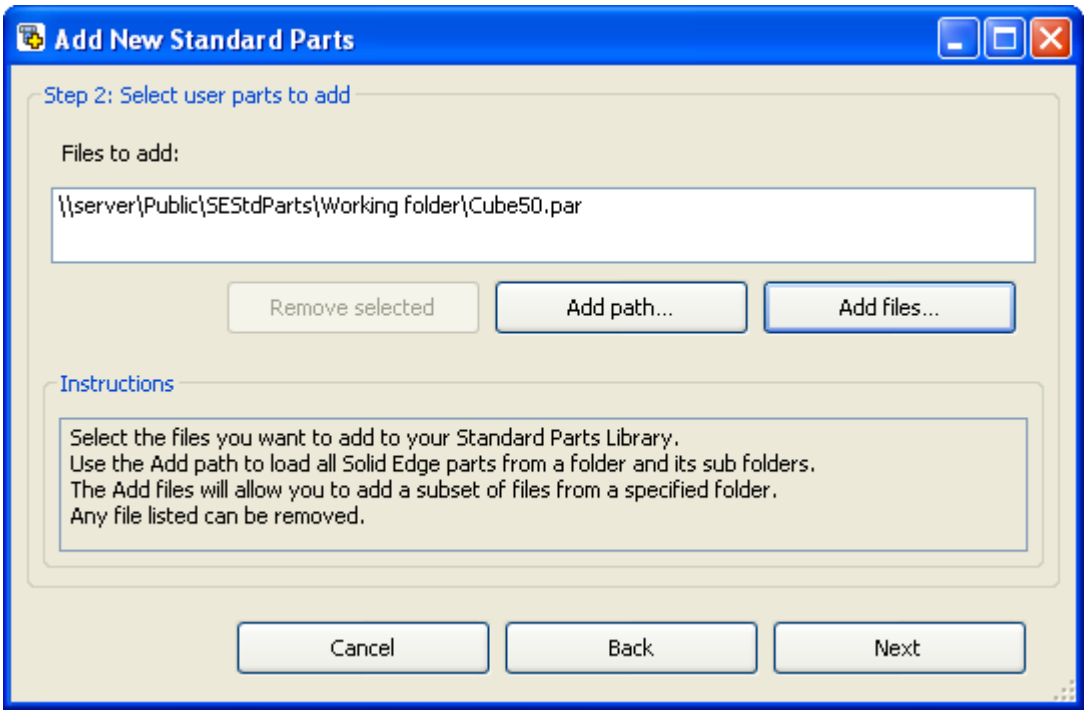
Step 1

Select "Add from manual data entry" and click on "Next".


Add New Standard Parts
Step 1: Specify the method for adding user parts
Add from Family Part Master
Add from active Microsoft Excel worksheet
Add from manual data entry
Instructions
You can add your own custom parts to the Standard Parts Library.
Choose one of the options to get started.
Any parts you intend on adding must first exist.
Cancel
Back
Next

Step 2

In this step you should select the parts you want to register as customer standard parts. You can add documents from a specified path in bulk (Add path) or add particular files (Add files). You can even remove files from the list (Files to add) via the "Remove selected" button.

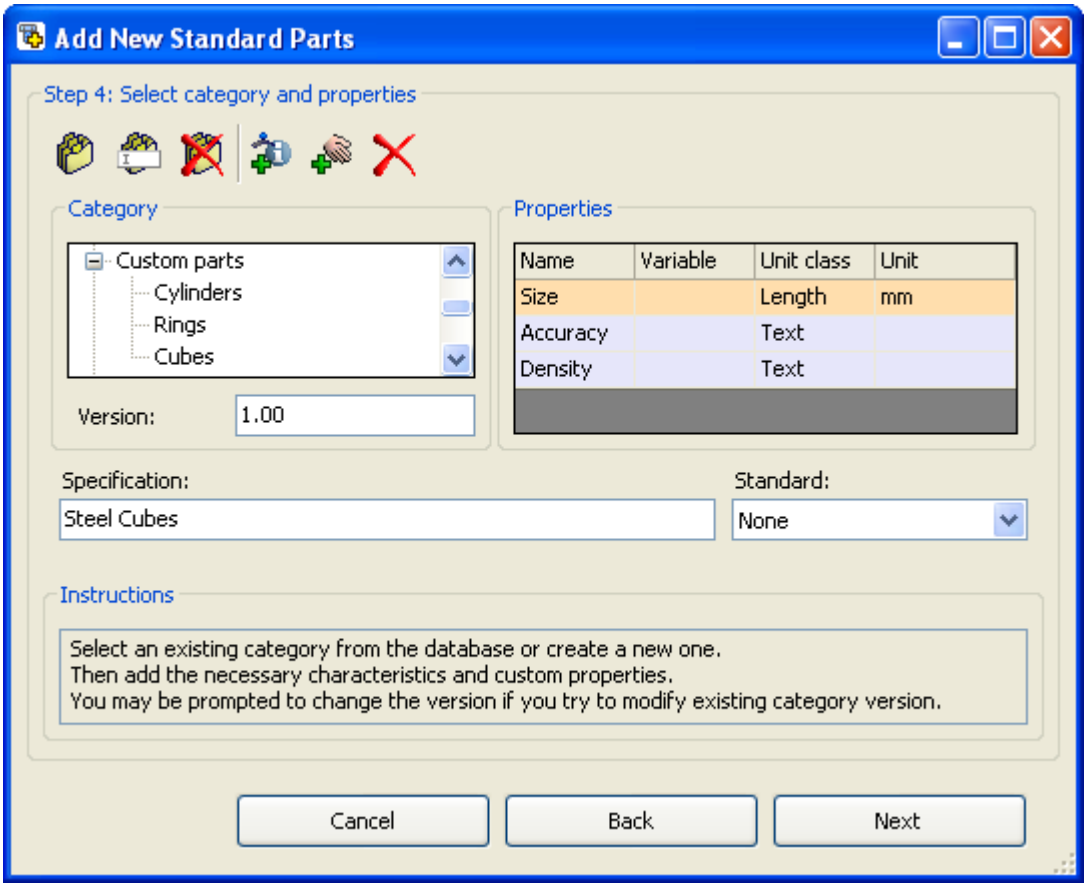

Add New Standard Parts
Step 2: Select user parts to add
Files to add:
\\server\Public\SEStdParts\Working folder\Cube50.par
Remove selected
Add path...
Add files...
Instructions
Select the files you want to add to your Standard Parts Library.
Use the Add path to load all Solid Edge parts from a folder and its sub folders.
The Add files will allow you to add a subset of files from a specified folder.
Any file listed can be removed.
Cancel
Back
Next

Step 3

Settings in this step are optional. If you want your custom part to be used as a fitting (or tube) by Solid Edge Piping you can check "Add to Piping" and set a number of coordinate systems pairs. To create a custom part to be used in Fastener Systems, check "Add to Fastener System" and select appropriate Fastener type (bolt, screw, washer or nut).

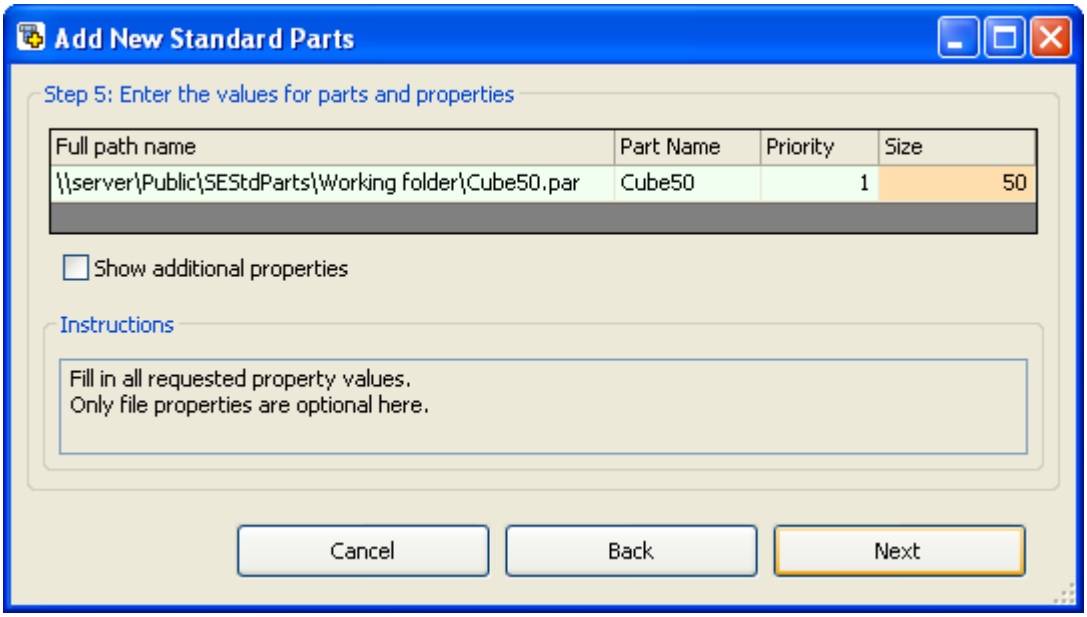
Step 4

In this step you should set a category for your custom part(s). Each category should have at least one characteristic assigned to it. If you want to add your custom parts to an existing category, you have to use the same characteristic set or change (typically increase) the category version. You can create two types of characteristics – constant or variable. For variable characteristics a variable must be set in the appropriate section of the Properties grid (this variable should correspond with an existing variable in the selected Family Master Part). For each characteristic you should set a unit class and the appropriate unit. In the Properties section there is also the possibility to add (or remove) custom properties. You can also enter a specification and set a standard for the created end category.



Step 5

This step requires, at a minimum, values for characteristics set in previous step to be entered. Full path, Part name and Priority are set automatically, but Part name and Priority can be modified manually.
If you check the "Show additional properties" box, then file properties (standard, Solid Edge and custom) will also be displayed in the grid. Filling out these values is optional.



Step 6

In this step it is possible to create a Visual Navigation category for your newly created category. This is the same as for creating customer standard parts from a family part master or using an Excel worksheet.

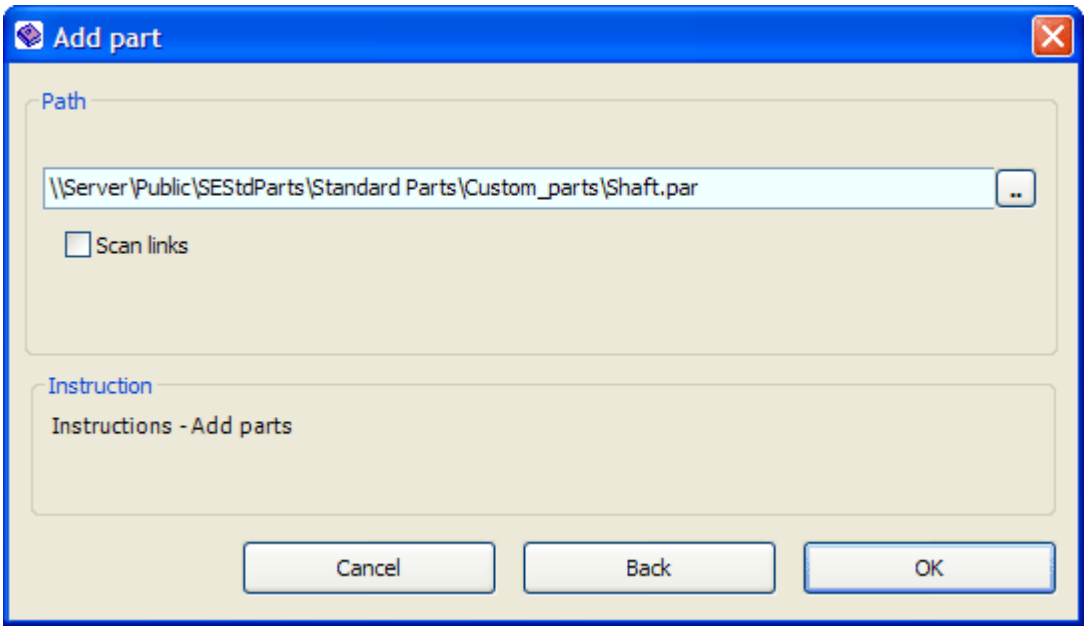
Note: If you are registering your custom parts into an existing end category, this step will be skipped automatically.

Step 7

In this step the necessary part information is created automatically at first. If you click on "Finish", the required parts will be copied to the Custom_parts folder (in the generated parts file location) and registered in the database.

Adding an Existing Single Part File

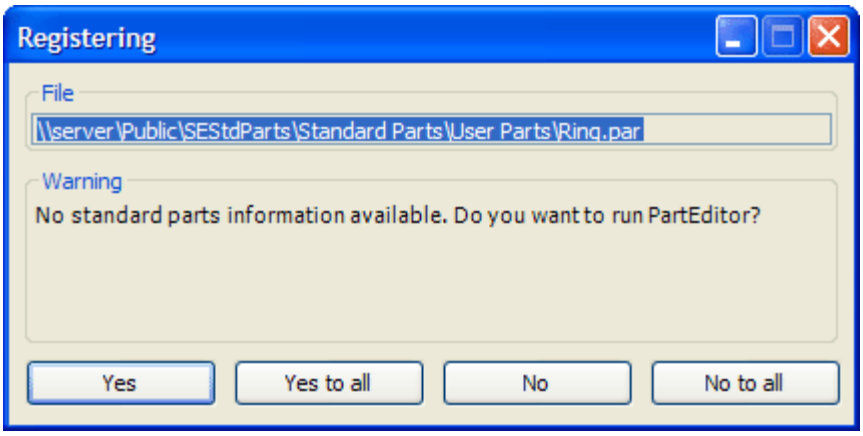
The "Add from existing single part file" option in Solid Edge Standard Parts Administrator makes it possible to add a content of a single file to the database.
In the Add part dialog you can browse (in the Parts or Generated parts location only) to a single file. It can be Single part, Parent part or any custom Solid Edge document. If the Scan links option is checked, links found in the selected part will be checked and linked documents will also be registered.



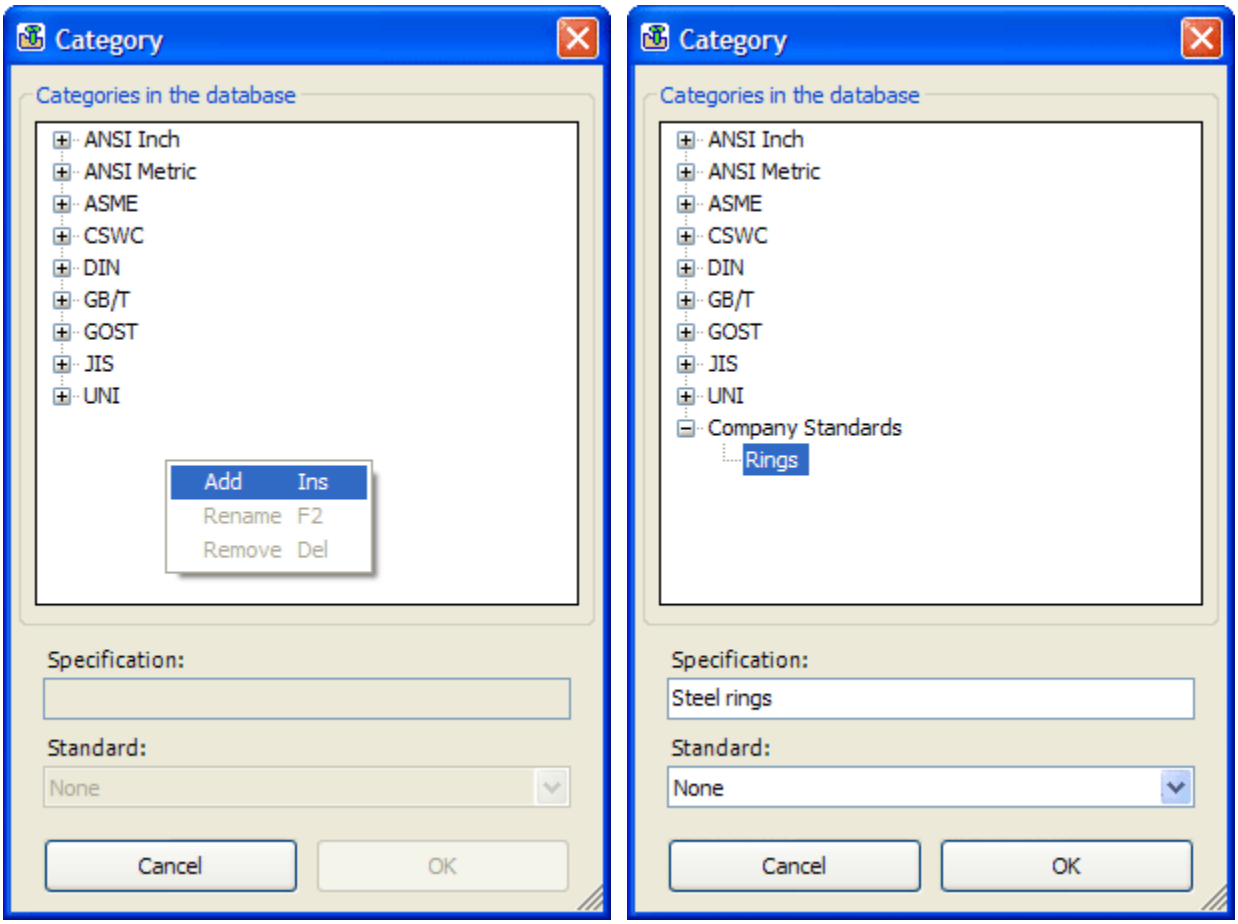
After clicking on "OK" the selected part will be registered.

Registering Non-standard Parts

If the selected part contains no standard parts information (category, characteristics etc.), the following dialog will appear:



1. Click on "Yes" to launch PartEditor.
2. In the PartEditor main form click on the "Category" button in the main menu to create a new category for your part.
    a. Click in the free space in the categories list with right mouse button to show the context menu.
    b. Click on "Add" to create new category (e.g. "Company Standards").
    c. Click with the right mouse button on the newly created category to add a subcategory (e.g. "Rings").
    d. Enter a specification for this "end category" (e.g. "Steel rings").
    e. Click on "OK"



3. You can also create a visual navigation category structure (optional). In the PartEditor main form click on the "Visual navigation category" button in the main menu to create a new visual navigation category for your part.
    a. Click on "Add node" to create new visual navigation node.
    b. Confirm the new visual navigation code.
    c. Set a name for the new node.
    d. Click on the "Change thumbnail" button and select an image to replace the visual navigation code in the visual category's thumbnail.

e. Change the category thumbnail for the end category if necessary (preview of the part is selected by default).

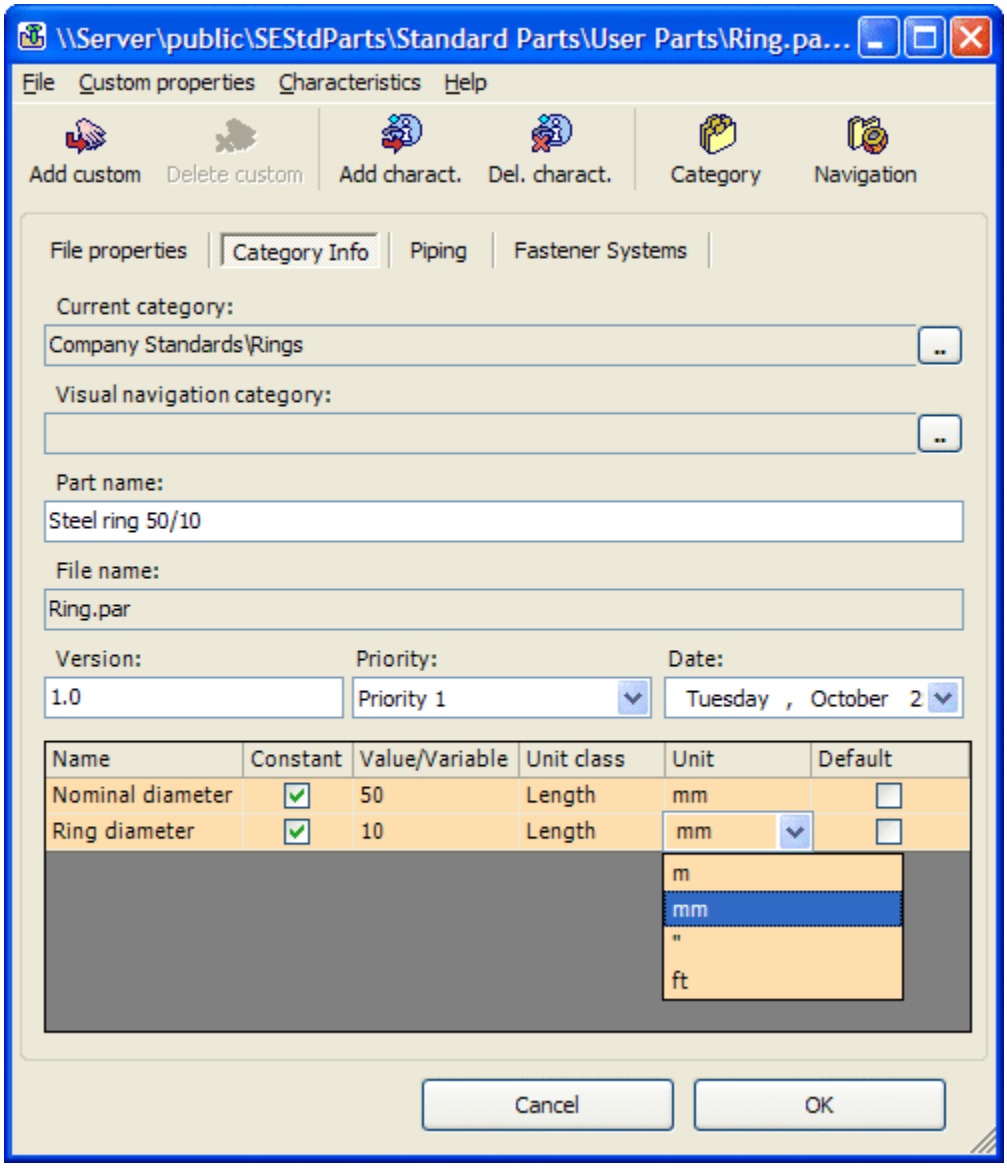
f. Click on "OK" to confirm your newly created visual navigation category.

4. Every Solid Edge Standard Part must have at least one characteristic[1]. These are shown in the bottom section of PartEditor's Category Info tab. For example, for defining a ring we need two characteristics (two diameters).

   a. Click on "Add characteristic".

   b. Specify names and values for all characteristics. Select unit class ("Length" in this case) and unit ("mm").

   c. Enter "Part name" (e.g. "Steel ring 50/10") and click "OK".

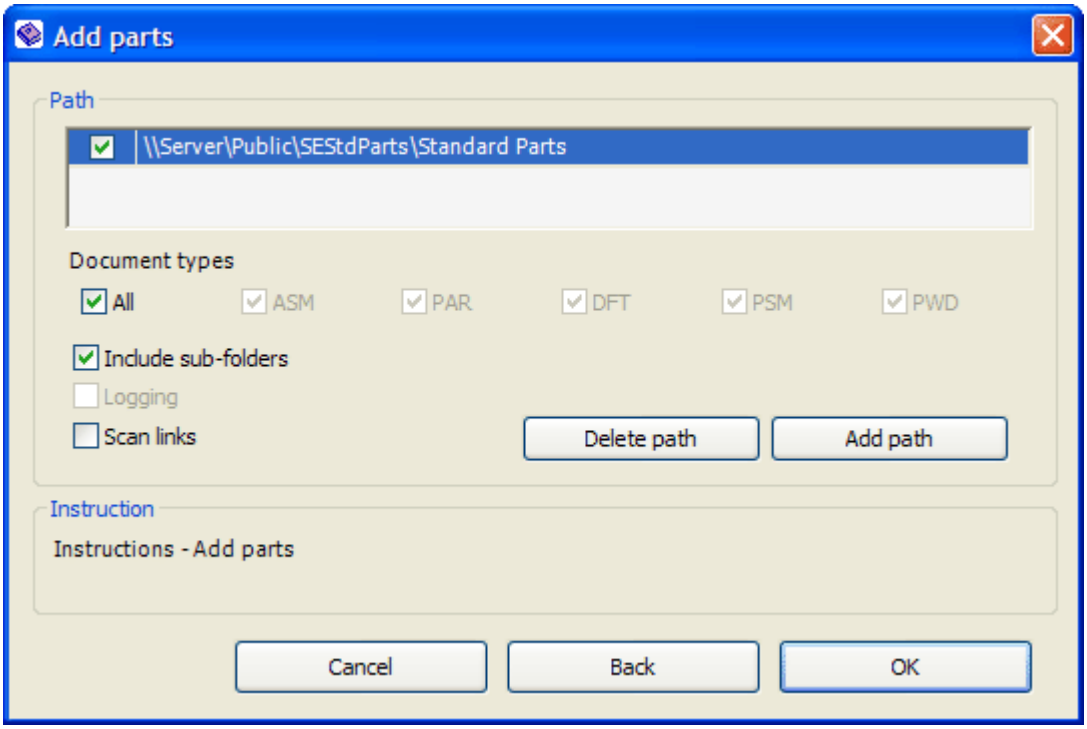


5. The user document "Ring.par" is now fully categorized as a standard part.

[1]Characteristic: key and probably unique property that belongs to described part (object)

Adding Existing Part Files

The "Add from existing multiple part files" option in Solid Edge Standard Parts Administrator makes it possible to register all Standard Parts (installed Libraries only) from the selected location(s). You can use it for registering the Solid Edge Standard Parts libraries.
For registering new parts check the required path or use the "Add path" button if it is not present in the list of paths (upper dialog section). Then click "OK" to launch the process.
There are also some additional options. You can select the document types that should be registered (all, ASM, PAR, DFT, PSM, PWD). If you check the "Include sub-folders" option the parts from the selected path/s and also from their subfolders will be registered. The "Logging" option enables logging information from the process to a log file, the log file must already exist and logging must be enabled. If the "Scan links" option is checked the links in parts will be checked during registration and linked parts will also be registered in the database.

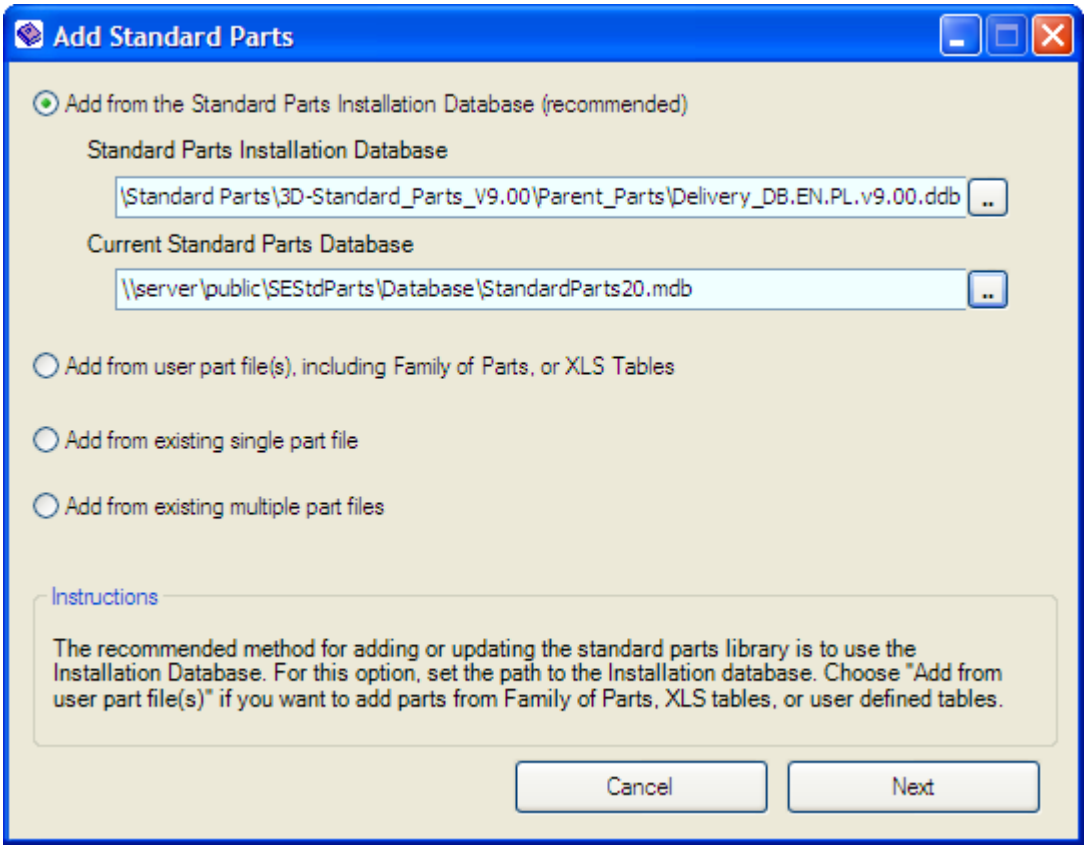


If a part from the selected path/s contains no standard parts information (category, characteristics etc.), then the "Registering" dialog will appear (see Adding an Existing Single Part chapter for more information).

Registering Standard Parts Using the Standard Parts Installation Database

Using an Installation Database (Add from the Standard Parts Installation Database) is the default method for registering standard parts contained in the Standard Parts Libraries. This technology has several benefits:

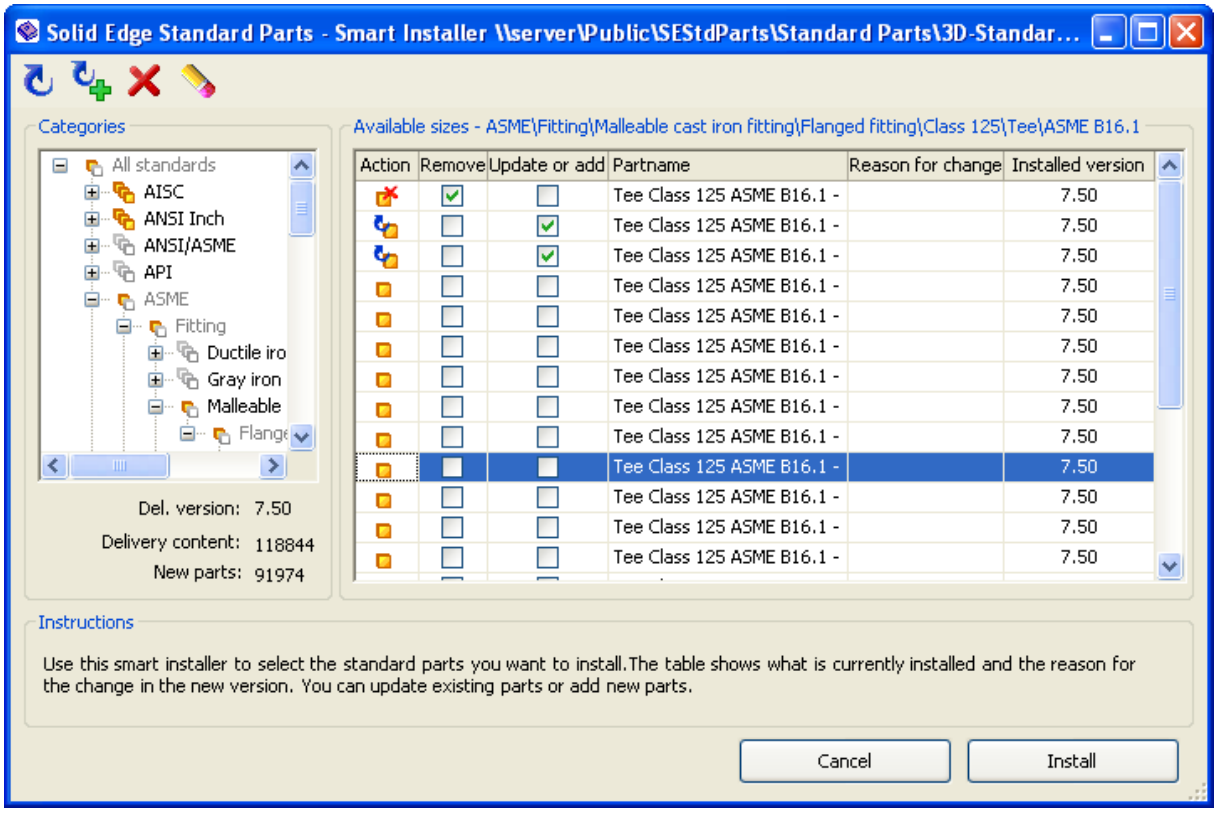
- A move from an older version of Solid Edge (V20 or lower) to ST is greatly simplified. It is possible to create a new database containing the same parts (as the original database) with updated versions.
- It is possible to select, from a clearly presented list (in Smart Installer), the parts you want to install. The benefit being that the parts you will not be using will not be registered in database. This can improve the performance of the Standard Parts application and shorten the time consuming process of registering standard parts in the database.
- If you obtain a new version of Standard Parts Library, then only the parts with changes (relative to the version you have installed) will be registered in the database.


Add Standard Parts
Add from the Standard Parts Installation Database (recommended)
Standard Parts Installation Database
\Standard Parts\3D-Standard_Parts_V9.00\Parent_Parts\Delivery_DB.EN.PL.v9.00.ddb
Current Standard Parts Database
\\server\public\SEStdParts\Database\StandardParts20.mdb
Add from user part file(s), including Family of Parts, or XLS Tables
Add from existing single part file
Add from existing multiple part files
Instructions
The recommended method for adding or updating the standard parts library is to use the Installation Database. For this option, set the path to the Installation database. Choose "Add from user part file(s)" if you want to add parts from Family of Parts, XLS tables, or user defined tables.
Cancel
Next

In the Add Standard Parts dialog there is the Database section where it is possible to set the input databases.
In the **Installation Database** field the Installation Database which will be used for registering/updating parts in the database should be selected. If there is an Installation Database located in the parts folder (but not in "Previews" or "Navigation" subfolder) it is selected automatically (Installation Databases are copied to the "Parent_Parts" folder automatically during the Standard Parts Libraries installation). If there are multiple Installation Databases, the one with the highest version is selected automatically.

Note: Installation Databases for Machinery and Piping Library are delivered separately. If you have purchased both of these libraries you should register/update in two steps and select one of the Installation Databases manually. Installation Databases can be distinguished via their file name, which contains "ML" for Machinery Library or "PL" for Piping Library (e.g. DeliveryDB.EN.ML.ddb and DeliveryDB.EN.PL.ddb).
The **Old database of parts** field serves to set a reference database. It has various purposes for various operations:

- **"Moving" from an older Solid Edge / Standard Parts version** – Suppose that you have newly installed Solid Edge and Standard Parts ST. However, you want to continue using your existing Standard Parts system created with a lower version. Given this scenario you can use an existing Top Level Standard Parts folder with one restriction: you should create new database. If you change the database via the Solid Edge Standard Parts Configuration Wizard (and save the changes), the Add Purchased Standard Parts dialog will pre-select the previously used database (lower version) as the "Old database of parts".
- **Updating a database with the new Standard Parts Library** – If you obtain a new version of the Standard Parts Library and try to add these parts via the Add parts dialog, then no "Old database of parts" will be displayed. In this case your current working database will be taken as reference database.
- **Registering parts in a new database** – There is also the case, that you would like to create a new database containing the same parts as an existing database, but you want to create the new database with up-to-date parts. In this case you can set your reference database (Old database of parts) manually.



Smart Installer Main Form

The Smart Installer main form consists of the sections "Categories" and "Available sizes".
The **Categories** section displays all the categories contained in the Installation and "Old" database. For each category there is a symbol describing the operation to be performed during installation:

All parts are currently installed and will not be updated

All parts are currently installed but will be removed from the database

Some parts are currently installed and no changes will be made

Some parts are currently installed but only installed parts will be updated

Some parts are installed and all parts will be added or updated

Some parts are currently installed but all parts will be removed from the database

No parts are installed and no parts will be added

No parts are installed but all will be added

These operations can be modified via the toolbar buttons **Update**, **Add and update All**, **Remove** and **Cancel actions**.
Under the category tree structure there is additional information displayed: Delivery version (version of delivered parts and Installation Database), Delivery content (total number of parts in Installation Database) and New parts (new in the Delivery version)
In the **Available sizes** section, single parts contained in the selected end category are listed. There are six columns displayed in the grid:

- The "Action" column displays information about the installation status of a single part:

  Part is currently installed and will not be updated

  Part is currently installed but will be updated

  Part is currently installed but will be removed from the database

  Part is not currently installed and will not be added

  Part is not currently installed but will be installed

- If you check the "Remove" box, the selected part will be removed from the database
- The "Update or add" checkbox allows updating or adding of a single part (available only when the selected part is contained in the Installation Database)
- "Part name" serves for part identification
- In the "Reason for change" column you can find information on changes made in the current version. Possible changes are:
  - Geometry – some changes have been made in the part geometry
  - File Name – file name of the parent and/or single part(s) has been changed
  - Part Name – part name has been changed
  - Classification – there were some changes made in the category information/structure
  - Fastener System – some information needed for Fastener Systems has been changed
  - Piping – some information needed for Solid Edge Piping has been changed
  - Invalid – part is no longer valid and will be removed from the database
  - Other changes – some unspecified changes have been made
- The "Installed version" column displays the version of the part that is currently registered in the database.

By clicking on the "Install" button all changes set in the Smart Installer main form will be performed on the current working database.

PartEditor

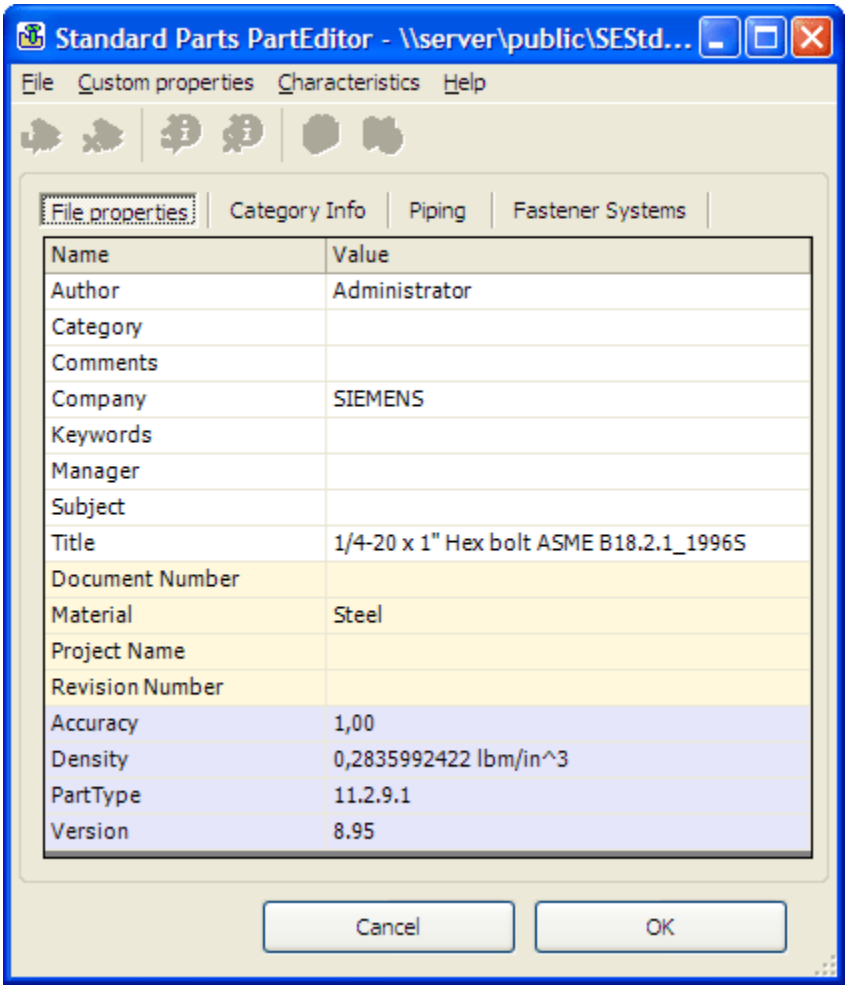
PartEditor allows you to edit the properties of documents registered in database. However there are several significant limitations:

- Non-generated parts cannot be edited. You can just view their properties.
- Generated standard parts cannot be edited because they are given a file status read-only. You can remove this status to enable editing, but it is not recommended to edit Solid Edge Standard Parts.
- If you want to add, remove or edit characteristics you must change the version (it is recommended to raise the version) or category of an edited part. The same applies for visual navigation.

From this you can deduce that PartEditor is mainly designed for creating and maintaining new customer standard parts.
There are four tabs in the main form: File properties, Category Info, Piping and Fastener Systems.

File Properties

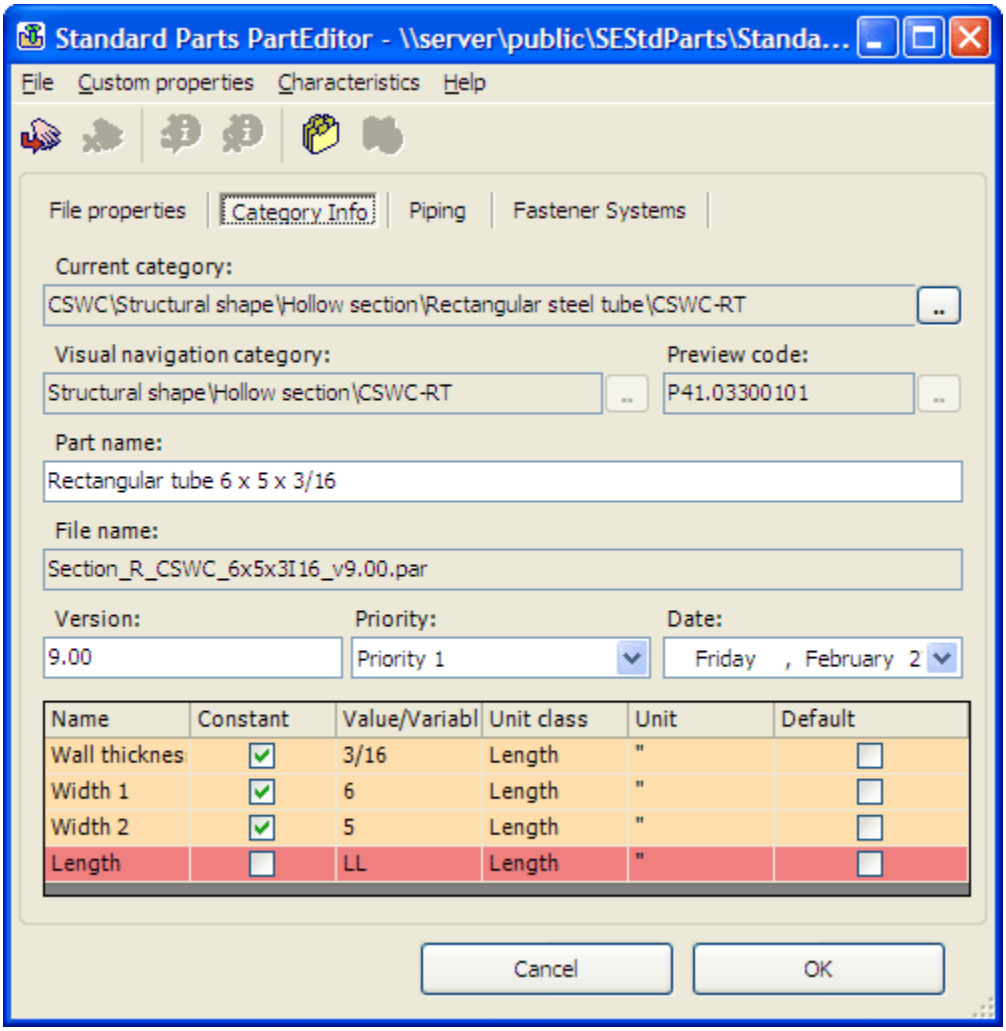
In this tab the standard file properties (Author, Comments, Keywords, Subject, Title, Category, Company, and Manager), Solid Edge properties (Material, Document Number, Revision Number, Project Name) and custom properties of the current document are displayed. You can edit the value of any property and you can even add or delete any custom property using the appropriate buttons (Add custom and Delete custom) in the toolbar.
If you want to use your customer standard parts with Solid Edge Piping, Fastener Systems or Frame Design you should add a custom property "PartType" with the appropriate value. You can find the list of correct values in the Appendix.



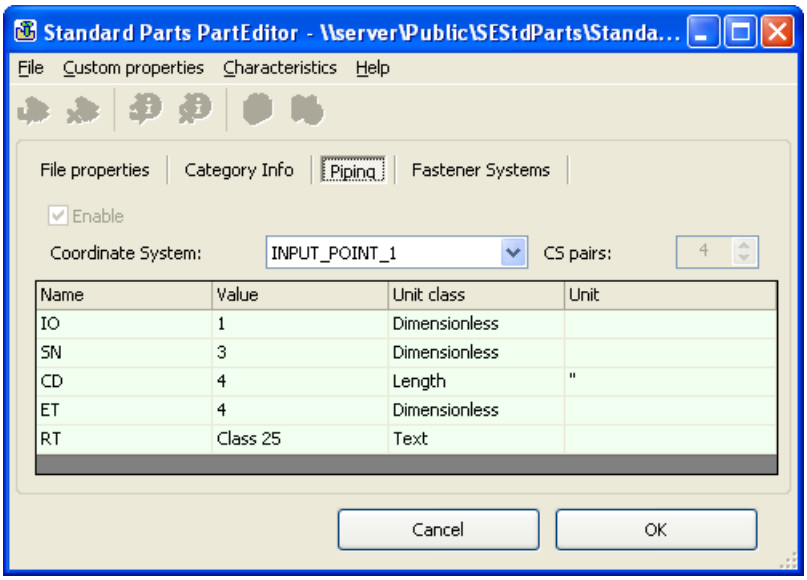
Category Info

The "Current category" box shows the full category structure (topmost category and subcategories including end category) of the current document. With the "Category" button or browse button you can call out the "Category" dialog and change the current category.
"Visual navigation category" displays a category structure for visual navigation. You can change it via the "Navigation" or browse buttons.
Part name is a global, language dependent identifier. It is displayed in the Solid Edge Standard Parts Administrator, in the Standard Parts dialog and it is also written to the file property selected in the Solid Edge Standard Parts Configuration Wizard. This value is not set in Customer standard Parts.
Version is what lets you keep your customer standard parts up-to-date. If you change the version you can edit the characteristics set (even add or delete characteristics) and also you can edit the visual navigation category. Characteristics and visual navigation must be the same for all parts with the same category and version.
Priority can be used for filtering documents in Standard Parts dialog. It must be enabled in Solid Edge Standard Parts Administrator.
The date that the customer standard part was created can be set for informational purposes.

Note: If you want to change names of the characteristics you should change the version first. Then you can delete the characteristics you want to rename and add new ones instead.



Piping

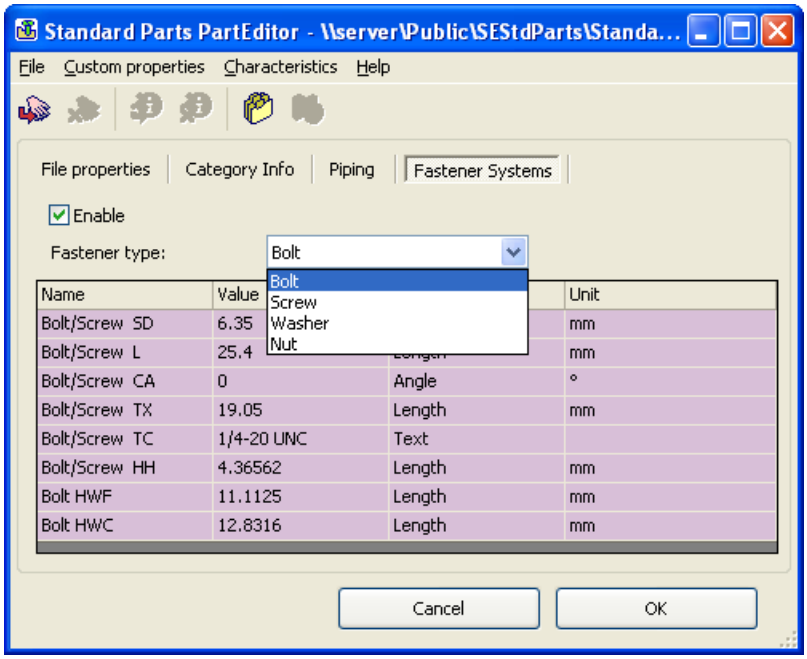


This tab displays the special property set that is needed for Solid Edge Piping. Here you can set the number of coordinate system pairs; you must then fill out the values in the table for each coordinate system.
For detailed information about Coordinate System parameters see Appendix.

For correct functionality with Solid Edge Piping, the components must fulfill the following conditions:

- The PartType custom property must be set correctly (see Appendix for details).
- The .par files of the pipe components have to contain the variables "d" = actual external diameter and "LL" = length.
- The fitting components have to be defined in the "PipeClass.txt" configuration file.

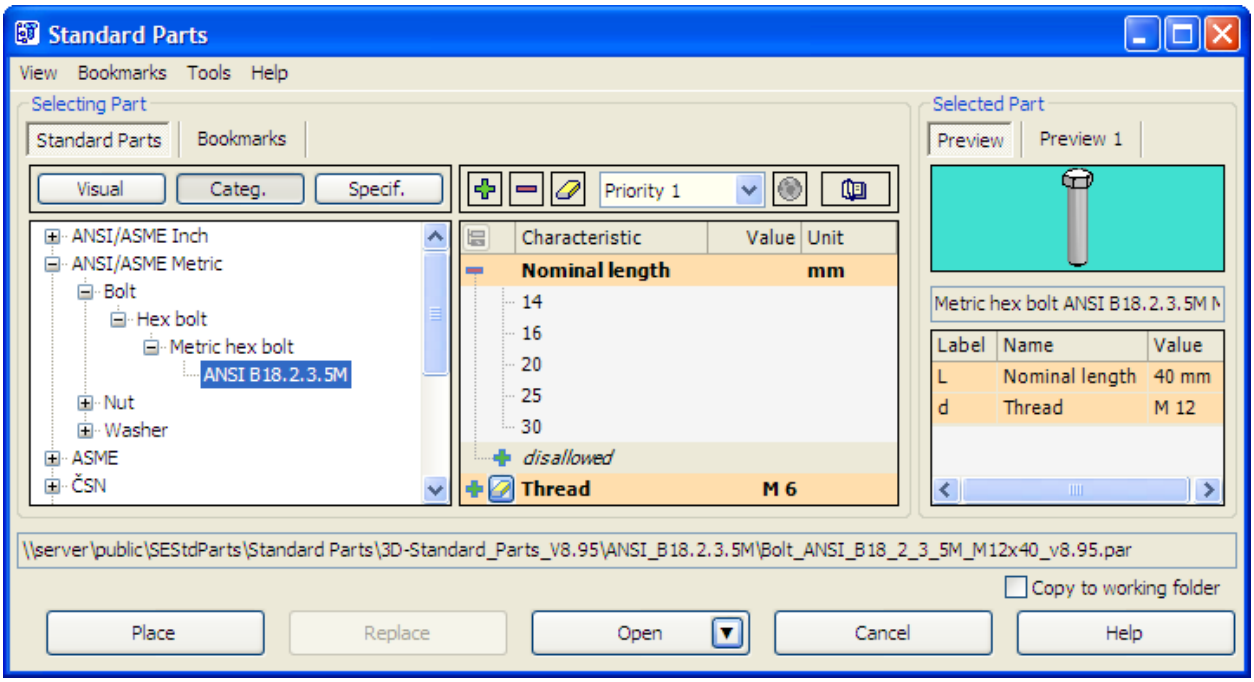
Fastener Systems



In this tab you should set the special properties which are required for Solid Edge Fastener Systems. If you choose the fastener type (bolt, screw, washer or nut) the table will be filled in with the appropriate properties. You should then enter only the values.
For detailed information about Fastener Systems parameters see Appendix.

Standard Parts

The Standard Parts dialog is a tool designed for easy selection of standard parts (and customer standard parts) which you want to place, replace or modify in Solid Edge assemblies. You can adjust the Standard Parts environment to best fit your needs. There are several possibilities to change the layout of the Standard Parts application form. You can simply switch between normal and vertical (similar to the Edge Bar) views using the F2 key. You can show or hide other panels such as the preview of the part (F3) and the parts properties (F4). See the following sections for a more detailed description.

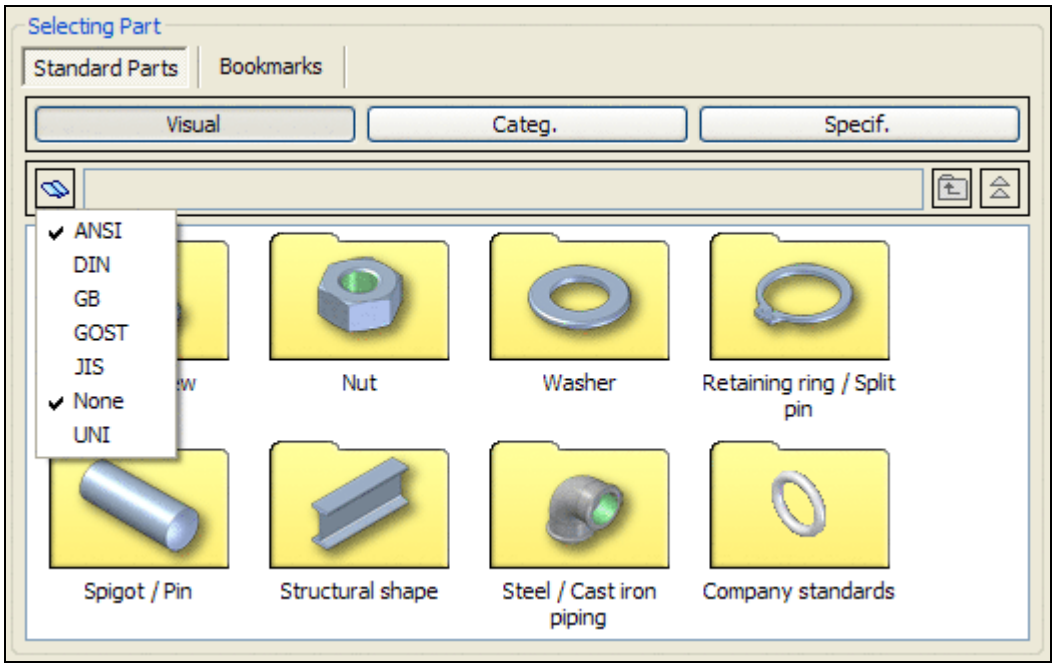


Selecting Part

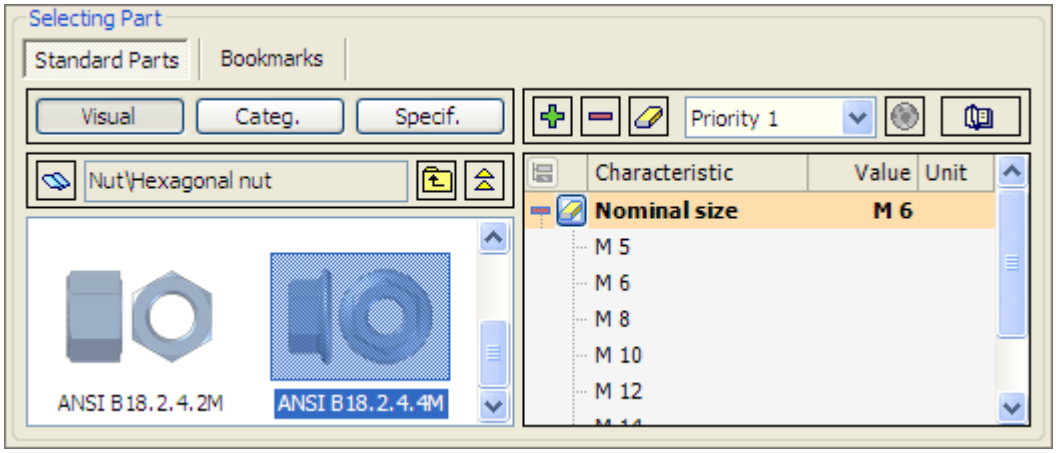
There are three ways to find a desired part. You can use visual navigation, the categories tree view or the specification view. You can also add selected parts as bookmarks for easier searching in the future.

Visual Navigation

Visual navigation makes it possible to find the desired part easily with pictograms which are specific to the type of the parts.

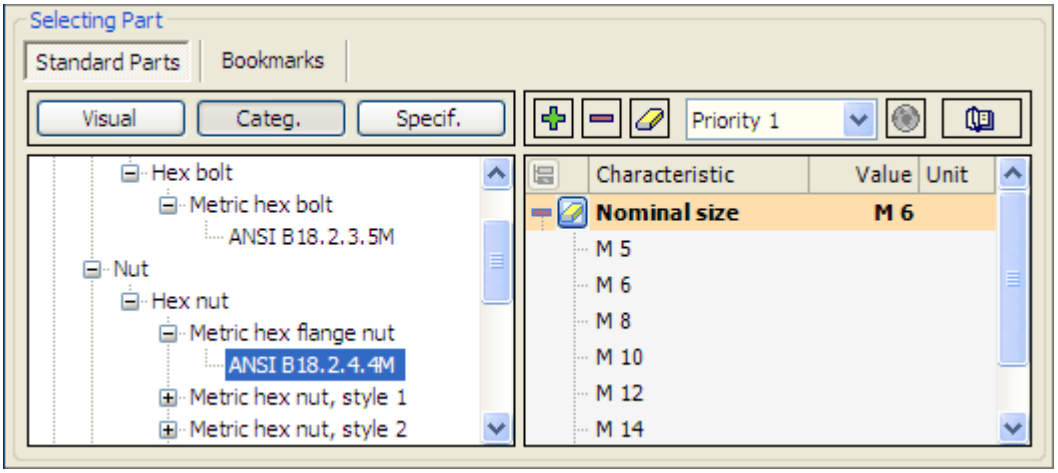


Visual navigation is active when the "Visual" button in the navigation selection bar is pressed. Visual categories are then shown in a section below. In the above image there are top level categories displayed. Therefore the "Level Up" and "Top Level" buttons (on the right above the categories display) are disabled. With the "Change selection of standards" button you can reduce the number of international standards to be displayed.
A double click on any category you will display its content. Categories which contain subcategories are displayed with yellow background (a folder image).
The end category has got no background (see image below). When you click on it the characteristics for the variants contained within will be displayed and you can select the desired variant.



Categories Tree View

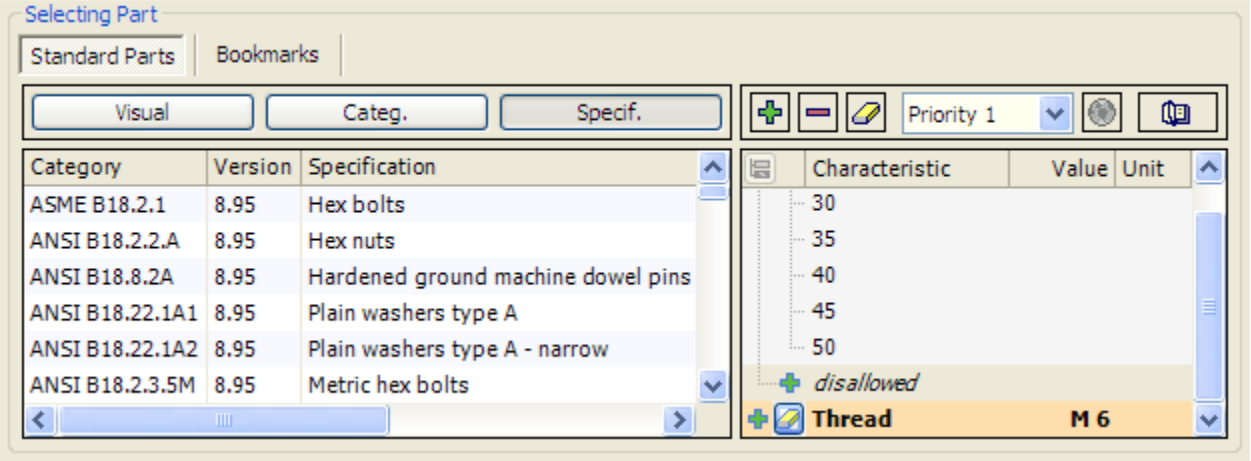
The concept of this view is very similar to visual navigation but the nodes are not displayed as images. A tree view is displayed with description of all categories. The key identifier (top-level node in the tree) is the international standard (e.g. "DIN").



When you click on an end category (a category without any subcategory) the characteristics for the variants contained within will be displayed and you can select the desired variant.

Specification View

If you select the specification view there is just a list of end categories (standards) with versions and specifications displayed. You can select the category which fits best to your needs. If you click on a category the characteristics for the variants contained within will be displayed, and you can then select the desired variant.



Characteristics

This section is shown in visual navigation, categories tree view and the specification view. It appears when an end category is selected and allows selection of a particular variant from the selected end category (see the right section of the image above).
All the characteristics for the currently selected end category and all their possible values are displayed. If you click on a value, it will be selected and the appropriate characteristic will collapse. The values of the other characteristics that are no longer available (in combination with selected value) are then disabled.
Selection is completed when values for all characteristics have been selected (all characteristics are collapsed with the selected values displayed). The selected part is then displayed in the 'Selected part' section and can be used for the desired operation.
There are several tools to aid selection:

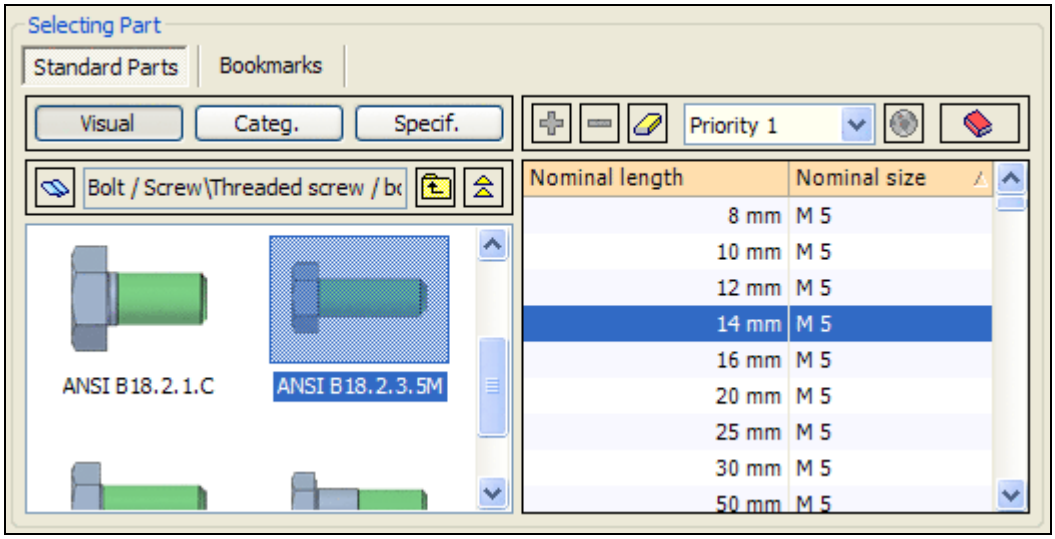
- Expand all characteristics expands all values for each characteristic. The current selection is kept.
- Collapse all characteristics collapses all values for each characteristic. No new values are selected in the process.
- Delete complete selection cancels the selection. Values of all characteristics are expanded.
- Priority filter allows selection of only those parts with priority higher or equal to the priority set in combo box.
- The open associated HTML or PDF file command displays a HTML or PDF file, typically containing some additional information about selected end category.
  There are two different ways to associate files with a category:
  a) Associated files must be located in HTML folder which is a subfolder of the Solid Edge Standard Parts Installation folder (e.g. "\\Server\Public\SEStdParts\HTML"). The file name must correspond with appropriate end category and must have 'htm', 'html' or 'pdf' extension, e.g. "ANSI B18.2.2.D.htm" (for the end category ANSI B18.2.2.D).
  b) It is also possible to add a "html.txt" file to the Solid Edge Standard Parts Installation folder. Associated file/s should be specified by name/s in row record/s with the following syntax: "last subcategory;file name".

Example: ANSI B18.2.2.D;\\server\documents\ANSI_nuts.pdf (for end category ANSI B18.2.2.D the ANSI_nuts.pdf file located in \\server\documents\ will be displayed)

- Show standard parts list changes the type of view (see next chapter).
- Show/hide alternative characteristics allows to display/hide alternative characteristics of the selected end category (if available). Alternative is such a characteristic which is not necessary or is redundant for unambiguous parts selection. It may be e.g. a size defined in inches when there is also the same size expressed in millimeters. Alternative characteristics are hidden by default; the reason is to simplify part selection.

Standard Parts List

This is an alternative possibility for displaying variants in an end category. This view replaces the default characteristic tree-list view for the selected standard (end category). The standard parts list does not show all the possible values of all characteristics, instead it just displays the available combinations. This option only makes sense for end categories with at least two characteristics.
You can change the display type by clicking on the book symbol in the top right of the "Selecting Part" section.

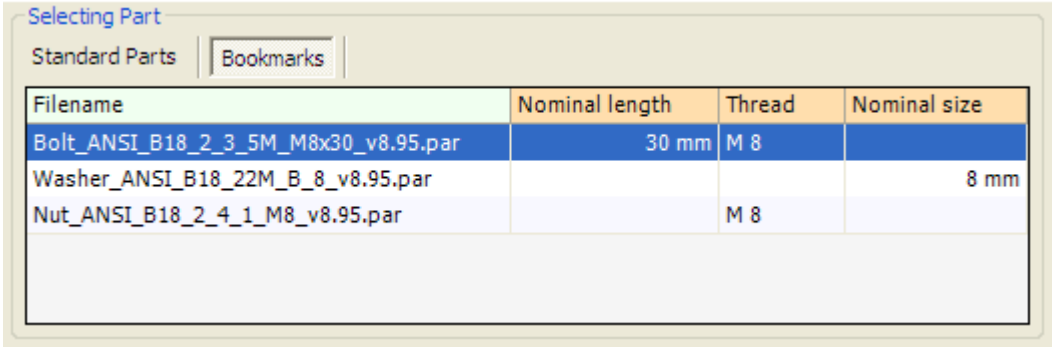


Some characteristics are always shown, but you can display other properties[1]. To do this, select "Display of parts list" in the "Tools" menu and then in the "Properties to display" dialog check the properties you want to add.

Bookmarks

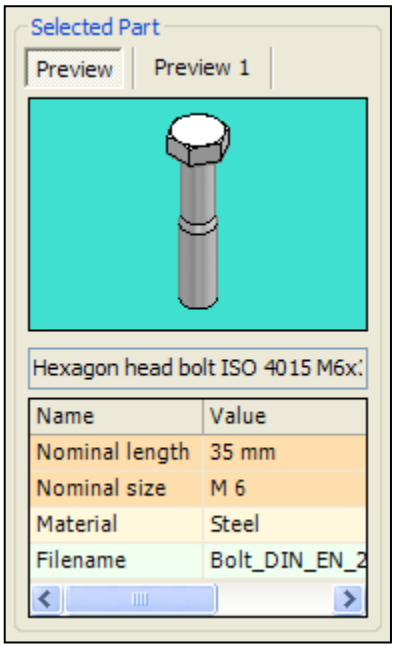
Using bookmarks is advantageous if you use certain parts over and over again. Adding a bookmark is very easy: when the desired part is selected just use the "Add Bookmark" function from the "Bookmarks menu.

[1] Meaning a property contained in the current database. If the property has no value for the selected part then it is displayed without a value.



The properties list displayed in the "Bookmarks" tab is the same as for the "Standard parts list". You can also change them in the same way (see "Standard Parts List section").

Selected Part



In this section of Standard Parts dialog you can see information about the selected part. The specifications, characteristics, selected properties and a preview are displayed.

Preview

At the top of this section, there is a preview of the selected part displayed. You can select the "CAD" preview (Preview) or "additional" (DXF or GIF) preview (e. g. Preview 1) if available.
In the Standard Parts options (Tools) you can set the type of CAD preview. It may be 3D (you can move and rotate the preview like in Solid Edge) or bitmap (the image contained in the part file).
3D preview can have two different background colors. Cyan (see figure) signifies a non-generated part. Dark blue is for generated parts.
Displaying the 3D preview may take time for more complicated parts or assemblies. If you want to increase performance, it is better to switch to the bitmap preview or disable the preview completely.
DXF or GIF preview usually shows the meaning of the different characteristics of the selected part. These previews are an advantage of Solid Edge Standard Parts Libraries.

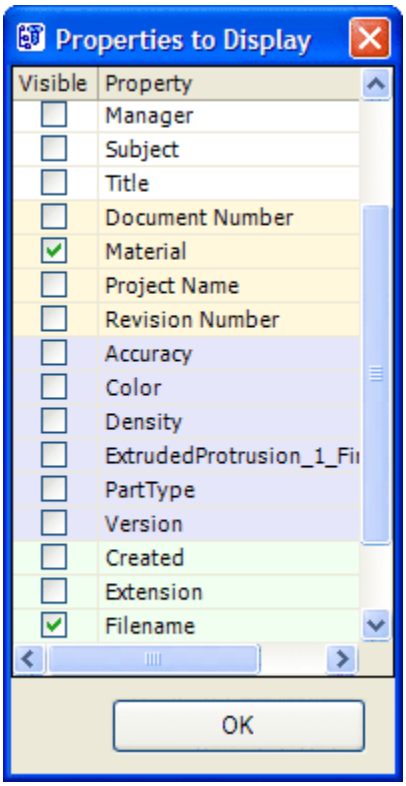
Specification

In the row below the preview the specification is displayed. This is a brief description of the selected part. The specification is sometimes called "Part name".

Displayed Properties

The bottom of the Selected Part section displays the characteristics and file properties of the selected part. Characteristics (including alternative ones) with their values are always shown; displaying of properties is optional. By clicking the right mouse button over this section you can open the "Properties to Display" dialog (see figure). You can then check the properties which you want to be displayed for each selected part.

Note: Only properties with non-empty values will be displayed.



Interaction with Solid Edge (CAD operations)

There are three possible operations you can perform with the selected part. These are "**Place**", "**Replace**" and "**Modify**".

Place

This command allows you to place selected part into the active Solid Edge assembly (traditional or synchronous). When no assembly is open, the place command is disabled.[1]

If you want to place a part:

1. Open the assembly where you want to place the part. If you are creating a new one – save it (e.g. to the Solid Edge Standard Parts working folder)
2. Launch the Standard Parts dialog from the Edge Bar / Part Library.
3. Select the part that you want to place.
4. Click on "Place".
5. Set appropriate relationships in the Solid Edge assembly (if necessary).

Note: If there were the same occurrences with different versions placed previously on the same assembly level, the user is given the chance (after finishing the place operation) to replace all the same occurrences with the current version.

Replace

If you want to replace a part, assembly, weldment or sheetmetal part in the Assembly environment with a standard (or customer standard) part you can use the "Replace" command. If there are multiple identical occurrences, Standard Parts asks you whether you want to replace all the same occurrences. If you have several components selected (see step 2), then you will be asked for each component separately.
If you want to replace a part:

1. Open the assembly (traditional or synchronous) where you want to replace a component.
2. Select the component (or components) that you want to replace.
3. Launch the Standard Parts dialog from the Edge Bar / Part Library.
4. Select the part with which you want to replace the original component.
5. Click on "Replace". (If it is disabled, press "F5" to synchronize with Solid Edge first.)

Modify

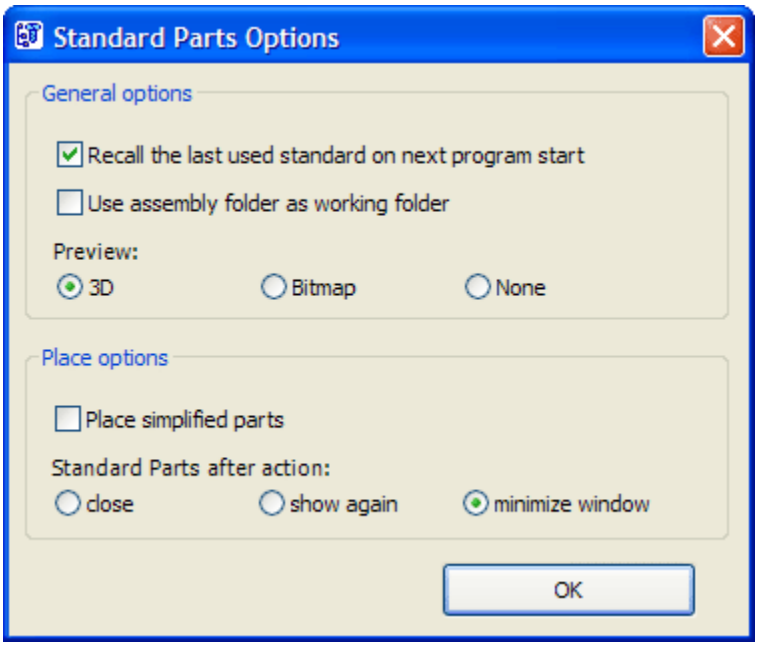
This command only supports variable parts. The modify command is not the same as replacing/placing the other parts in an assembly; the current part only has a variable characteristic changed. The modify operation is useful for parts such as profiles, pipes or pins – the user can change the value of a variable characteristic (e.g. "Length") over and over again. A typical situation is adjusting the length of a profile.

If you want to modify a part:

1. Open the assembly (traditional or synchronous) containing the part you want to modify.
2. Select the variable part you want to modify (it must be currently registered in the database and contain at least one variable characteristic[2]!).
3. Launch the Standard Parts dialog.
4. Press "F5" to synchronize with Solid Edge.
5. Click on "Modify".
6. Set new values for the variable characteristic/s and click on "OK".

[1] Some operations are also disabled when the user has no Standard Parts Library license.
[2] Variable characteristics are displayed in red in the Standard Parts dialog.

Standard Parts Options



In Standard Parts Options dialog you can control the behavior of the Standard Parts dialog and the properties of the part which is being viewed or placed.
"**Recall the last used standard on next program start**" sets whether the last selected end category will be selected again the next time Standard Parts is started.
If the "**Use assembly folder as working folder**" option is checked then parts are copied to the location where the assembly is saved, instead of the working folder. This option is only used if the "Copy to working folder" option is set in the Standard Parts dialog, or for variable parts which are always copied to the working folder.
Selection of the "**Preview**" type controls the CAD preview of the selected part in the Standard Parts dialog (see the Selected Parts section for more details).
Checking the "**Place simplified parts**" can increase performance, especially for large assemblies.
With the "**Standard Parts after action**" switch you can control what the Standard Parts dialog should do after finishing an action (place, replace, modify). It may be closed, be shown again or be minimized.

Cooperation with Teamcenter

The cooperation of Solid Edge Standard Parts and Teamcenter brings some specifics. These concern mainly managing Teamcenter identifiers and Part names of the standard parts.

Teamcenter IDs

Teamcenter identifiers are assigned to standard parts on uploading to Teamcenter. This happens either on standard parts generation process or when using the Migrate to Teamcenter command for already generated parts. It does not matter if the part is generated from Solid Edge Standard Parts Administrator or if it is being opened/placed from the Standard Parts application.
Teamcenter identifiers Item ID and Revision are stored in properties Document Number and Revision Number of the generated parts respectively. Although these identifiers are normally assigned automatically by Teamcenter, there is also a possibility to use preset "custom" values. If both Document Number and Revision Number properties are set before uploading to Teamcenter (generating or migration), their values are preserved and assigned to Teamcenter identifiers. If any of these properties is empty before uploading, both Teamcenter IDs are automatically assigned and stored to the properties (original values are overwritten in this case!).

Note: File properties of non-generated parts can be set using the Edit File Properties command in Solid Edge Standard Parts Administrator.

Project Number Property

When working with Teamcenter, the Project Number property of the standard part files can be managed in various ways. If the value of this property is predefined before generating part, this value is preserved in the generated part. If it is not predefined, Project Number will get the same value as Document Number (Teamcenter Item ID). When Project Number is set in Solid Edge Standard Parts Configuration Wizard as the target property where Part names should be written, Project Number will be filled with Part name.

Note: The Project Number value can be shortened automatically when it exceeds the limit of 32 or 128 characters (depends on Teamcenter setting "TC_Allow_Longer_ID_Name"). A message box is displayed in such situation.

Troubleshooting

This article summarizes known problems which can appear when using Solid Edge Standard Parts applications and describes how to solve them.

1. **Solid Edge Standard Parts or Solid Edge Standard Parts Administrator does not display available parts within the selected end category.**
   This problem can have two reasons. The first one is that the database file has the read-only status or the write access permission is not set for the current user. Just permit the write access for all the users to solve the problem.
   The second reason could be database instability. When the limit size (1GB) of the Microsoft Access database is exceeded, it may become unstable. To fix this problem you should use the Compact Database command available in Solid Edge Standard Parts Administrator (see the Compact Database Section for more details).

2. **Solid Edge Standard Parts Administrator cannot be launched when Teamcenter is used.**
   This problem appears when the Solid Edge Embedded Client is not installed. Install it to fix the problem.

3. **Cannot connect to Solid Edge application database.**
   This message can appear on Solid Edge Standard Parts or Solid Edge Standard Parts Administrator start. Similar messages can appear on saving configuration via Solid Edge Standard Parts Configuration Wizard.
   This means that the Solid Edge application database (*.ugs) referenced in the Standard Parts database is not available on its original location (the location where the database was created).
   Typical situation how it can happen is when a user installs Standard Parts (including Part files) locally and then moves the whole Standard Parts folder structure to a network location.
   This problem can be fixed by creating a new database.
   The other possibility is to edit the Standard Parts database file (*.mdb) using the Microsoft Access application. Correct path to Solid Edge application database should be set in the "DBConfig" table.

4. **English filenames instead of localized part names are displayed in the Edge Bar.**
   This behavior is as designed (it was changed in some SP for Solid Edge version 19 and 20). If you want to change it to the state known from previous versions, you should add the following record into the registry:
   [HKEY_LOCAL_MACHINE\SOFTWARE\Unigraphics Solutions\Standard Parts]
   "EnableOccNameOverride"="1"

Appendix

Coordinate System Parameters

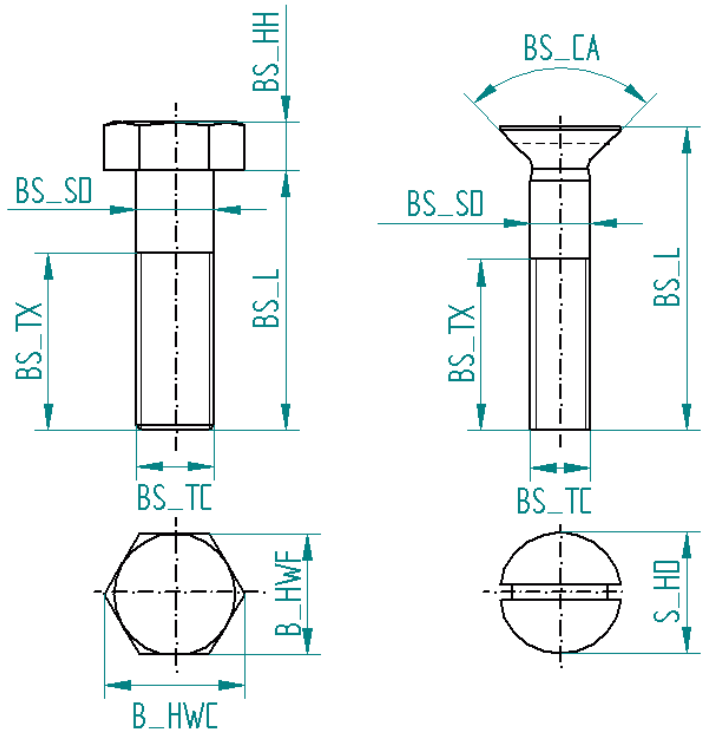
Name	Description	Value or Special Instructions
IO	Type of CS (Coordinate System)	1 = input cs 2 = output cs
SN	Sequence Number	Pipes have two pairs of CS with SN = 1 and 2. The sequence numbers can be assigned to the pair INPUT/OUTPUT_POINT_1 or INPUT/OUTPUT_POINT_2 arbitrarily.
		In fittings with two openings, SN = 1 is the pair at the opening with bigger diameter, SN = 2 is the opposite pair. If the openings are the same size, SN = 1 is the pair at the opening with male connection and SN = 2 is the pair at the opening with female connection. If the fitting has no male/female connections, the sequence numbers are arbitrary.
		Laterals have SN = 3, the pair at the lateral opening, SN = 1 is the pair at the opening near lateral opening and SN = 2 is the pair opposite to pair with SN = 1.
		T's have SN = 3 for the side opening pair, SN = 1 is opening with bigger diameter and SN = 2 is the pair opposite to pair with SN = 1.
		Y's have SN = 1 for the pair at the opening opposite to both branches which get SN = 2 and SN = 3.
		Crosses have SN = 1 is the pair at the opening with biggest diameter, SN = 2 is the pair opposite to pair with SN = 1, SN = 3 is the pair at the opening with bigger diameter from the remaining openings and SN = 4 is the last pair.
CD	Connecting (nominal) diameter of the opening	
ET	End Treatment	0 = Not specified 1 = Threaded 2 = Socket Weld 3 = Butt Weld 4 = Flanged 5 = Welded Neck 6 = Blind 7 = Slip on 8 = Lap joint 9 = Socket 10 = Butt Weld - branch
RT	Rating of the part	

Fastener Systems Parameters

Bolt/Screw

Name	Abbrev	Description
Bolt/Screw SD	BS_SD	Bolt/Screw Shank Diameter
Bolt/Screw L	BS_L	Bolt/Screw Length
Bolt/Screw CA	BS_CA	Bolt/Screw Countersink Angle
Bolt/Screw TX	BS_TX	Bolt/Screw Thread Extent
Bolt/Screw TC	BS_TC	Bolt/Screw Thread Count*
Bolt/Screw HH	BS_HH	Bolt/Screw Head Height
Bolt HWF	B_HWF	Bolt Head Width Across Flats
Bolt HWC	B_HWC	Bolt Head Width Across Corners
Screw HD	S_HD	Screw Head Diameter

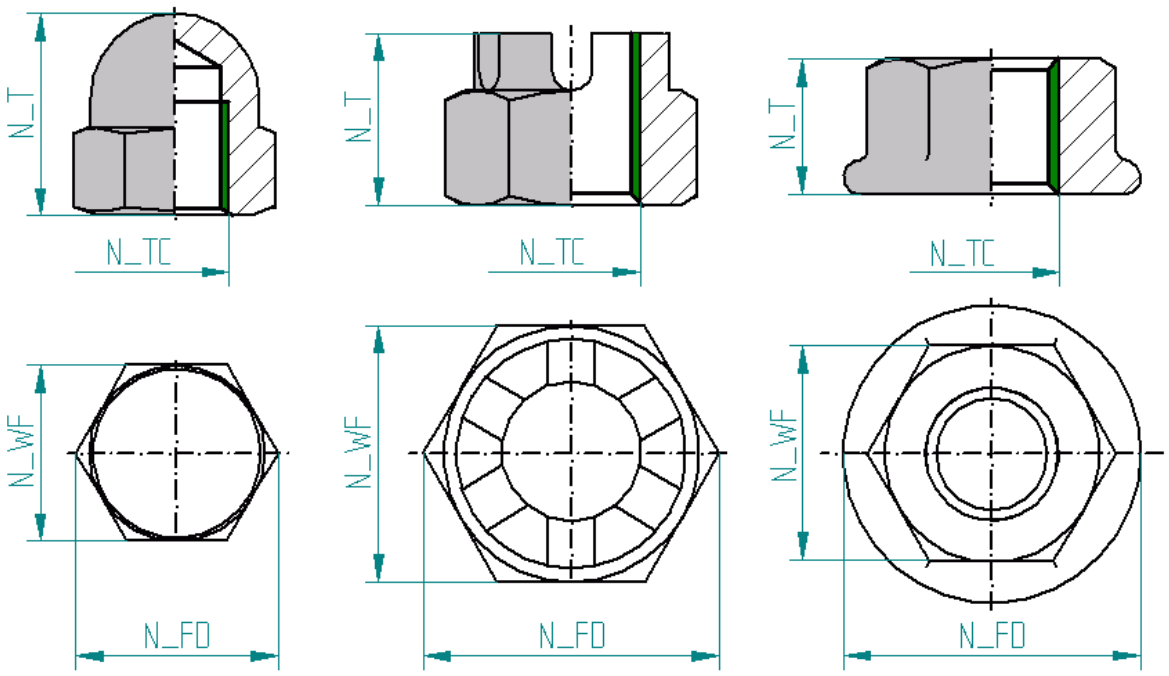
* *thread name listed in Holes.txt*



Nut

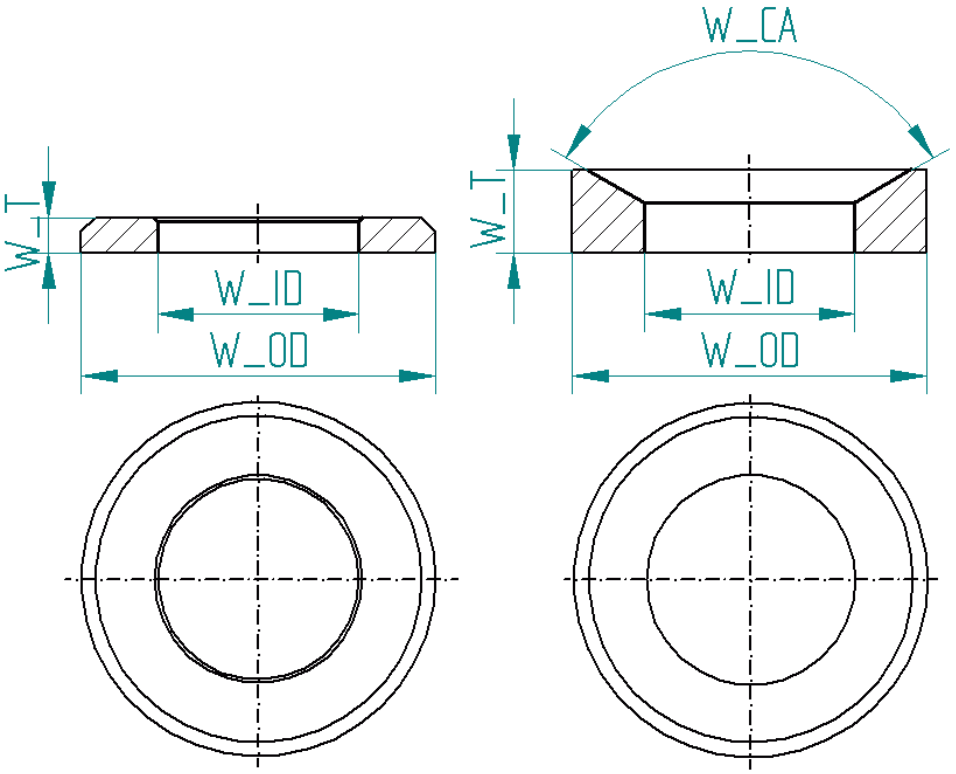
Name	Abbrev	Description
Nut T	N_T	Nut Thickness
Nut TC	N_TC	Nut Thread Count*
Nut WF	N_WF	Nut Width Across Flats
Nut FD	N_FD	Nut Flange Diameter

* *thread name listed in Holes.txt*



Washer

Name	Abbrev	Description
Washer ID	W_ID	Washer Inner Diameter
Washer T	W_T	Washer Thickness
Washer OD	W_OD	Washer Outer Diameter
Washer CA	W_CA	Washer Countersink Angle



Values of custom property PartType

Piping Components

1	Pipes	
2	Fittings	
	2.1	Elbows
		2.1.1 = 5 5/8° Elbows 2.1.2 = 11 1/4° Elbows 2.1.3 = 22 1/2° Elbows 2.1.4 = 30° Elbows 2.1.5 = 45° Elbows 2.1.6 = 60° Elbows 2.1.7 = 90° Elbows 2.1.8 = Side-outlet 90° Elbows 2.1.9 = Double-branch Elbows
	2.2	Bends
		2.2.1 = 11 1/4° Bends 2.2.2 = 22 1/2° Bends 2.2.3 = 45° Bends 2.2.4 = 90° Bends 2.2.5 = Return Bends 2.2.6 = 5 5/8° Bends
	2.3	Returns (180°)
	2.4	Y's
		2.4.1 = True Y's (90°) 2.4.2 = Straight Y branch
	2.5	Tees
		2.5.1 = Straight Tee 2.5.2 = Side Outlet
	2.6	Crosses
	2.7	Laterals
		2.7.1 = 45° Laterals
	2.8	Reducers
		2.8.1 = Concentric Reducer 2.8.2 = Eccentric Reducer
	2.9	Couplings
	2.10	Plugs
	2.11	Ends
	2.12	Caps
	2.13	Bushings
	2.14	Long Screws
	2.15	Nipples
	2.16	Sockets
	2.17	Collars
	2.18	Branch outlets
3	Flanges	
4	Unions	
5	Accessories	
6	Seals	

Fasteners

Codes for Element (All Fasteners)	
11	Bolts*
12	Screws
13	Washers**
14	Nuts
15	Retaining elements
16	Rivets

** PartType for Screws/Bolts is composed of four fields delimited by a period: <Element>.<Contact face>.<Head Type>.<Screw drive type>*
*** PartType for Washers is composed of two fields delimited by a period: <Element>.<Contact face>*

Screw/Bolt Contact Face	
1	None
2	Plain
3	82° countersunk
4	90° countersunk
5	100° countersunk
6	114° countersunk
Screw/Bolt Head Type	
1	None
2	Binding head
3	Binding head undercut
4	Button head
5	Eye head
6	Fillister head
7	Flat head
8	Flat head undercut
9	Hexagon head
10	Hexagon flanged head
11	Hexagon washer head
12	Cheese head
13	Indented hexagon head
14	Indented hexagon flanged head
15	Indented hexagon washer head
16	Knurled
17	Oval head
18	Oval head undercut
19	Pan head
20	Ring head
21	Round head
22	Square head
23	Square shoulder head
24	T head
25	Triangle head
26	Truss head
27	Socket head
Screw/Bolt Drive Type	
1	None
2	Hexagon socket
3	One-way

4	Phillips
5	Phillips square combo
6	Pozidriv
7	Slotted
8	Slotted Phillips combo
9	Spanner
10	Square socket
11	Torq-Set
12	Torx
13	Triangle socket
14	Tri-Wing
15	Four lobe
16	Six lobe

Washer Contact Face	
1	Plain symmetrical
2	Asymmetrical (spring, locks)
3	120° countersunk
4	90° countersunk

Nuts	
14.1	Cap
14.2	Castle nut
14.3	Hexagon flange nut
14.4	Hexagon nut
14.5	Octagon nut
14.6	Ring nut
14.7	Round knurled nut
14.8	Square nut
14.9	Lifting eye nut
14.10	Lock nut
14.11	Slotted round nut
14.12	Self locking counter nut
14.13	Pipe nut
14.14	Hex slotted hex

Profiles

Solid Shapes	
21.1	Square steel
21.2	Rectangular steel
21.3	Round steel
21.4	Half-round steel
21.5	Half-oval steel
21.6	Hexagon steel

Tubes	
22.1	Square tube
22.2	Rectangular tube
22.3	Round tube (pipe)

Structural Sections	
23.1	Equal leg angle
23.2	Unequal leg angle
23.3	Z-section
23.4	T-section
23.5	Channel section
23.6	I-section
23.7	Lipped channel section
23.8	Hat section
23.9	Lipped Z-section
23.10	Unequal channel section